QuakeSim Portal User's Guide

Contacts: Marlon Pierce marpierc@indiana.edu 812-856-1212
Jay Parker Jay.Parker@jpl.nasa.gov 818-354-6790
Peggi Li Peggy.Li@jpl.nasa.gov 818-354-1341

Overview

The QuakeSim portal and test bed consists of the following pieces:

1. A portlet-based portal environment that allows users to customize their displays and service interfaces.
2. Support for the QuakeSim codes Disloc, Simplex/Geofit, GeoFEST, VirtualCalifornia, Mesh Generation (Apollo/Akira), SLIDER, Phase Dynamical Probability (PDPC), Karhunen-Loeve Space-Time Pattern Analysis, Genetic Algorithm Analysis codes, and Hidden Markov Model Codes (RDAHMM).
3. XML based Web services that allow the user to
   - Transfer files from desktop to backend and between backend resources
   - Submit jobs
   - Monitor job progress on backend resources

- Archive and resubmit old jobs
- Visualize outputs
- Download output data
- Provide both human and application access to various databases
- Manage portal applications.

The general architecture is shown in Figure 1. For the test bed, we use the following resources:

- Complexity, a Sun Sunfire 880 server, acts as the main web server, runs the portal, and manages connections to remote services through client stubs.
- The fault database is hosted at USC.
- Services for job submission, file management, and job monitoring are hosted on a range of Linux and Solaris servers (danube, kamet, and grids).
- Complexity also hosts several local services for session management.
- Users may access other resources through SSH wrappers.

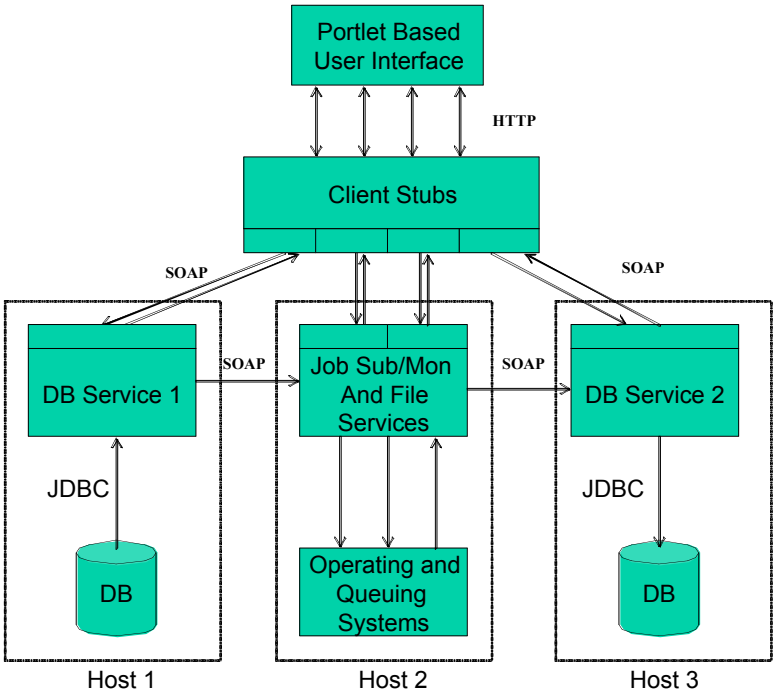

Figure 1 QuakeSim Architecture

Getting a Portal Account

First time users may request an account using the following steps.

1. Go to the portal URL, http://complexity.ucs.indiana.edu:8282/jetspeed. You should see the screen in Figure 2.
2. Click the "Create New Account" link in the upper right hand corner. You should see the screen in Figure 3.
3. Fill out the form and submit. You MUST use the correct email address: you will be notified by email when your account has been activated.

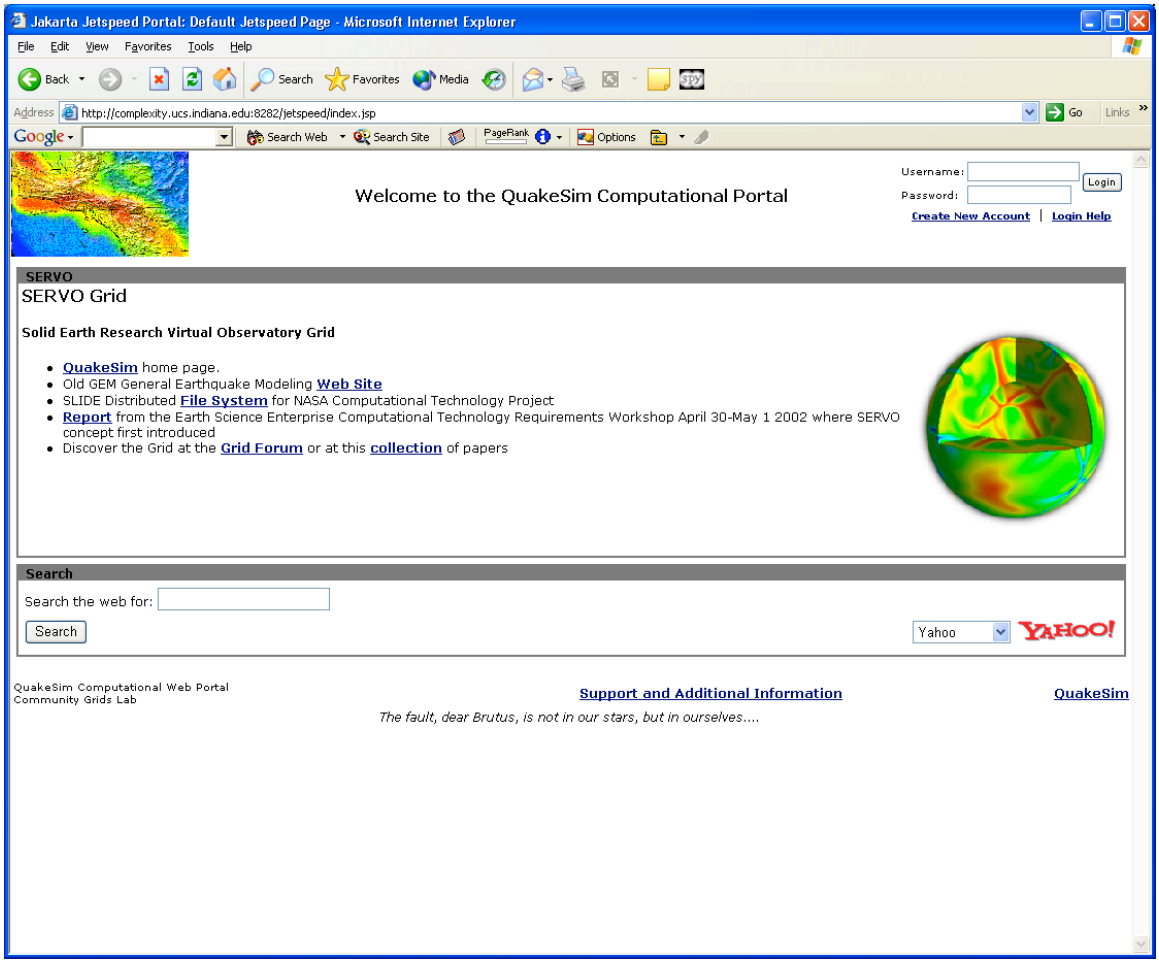

Figure 2 QuakeSim portal entry page

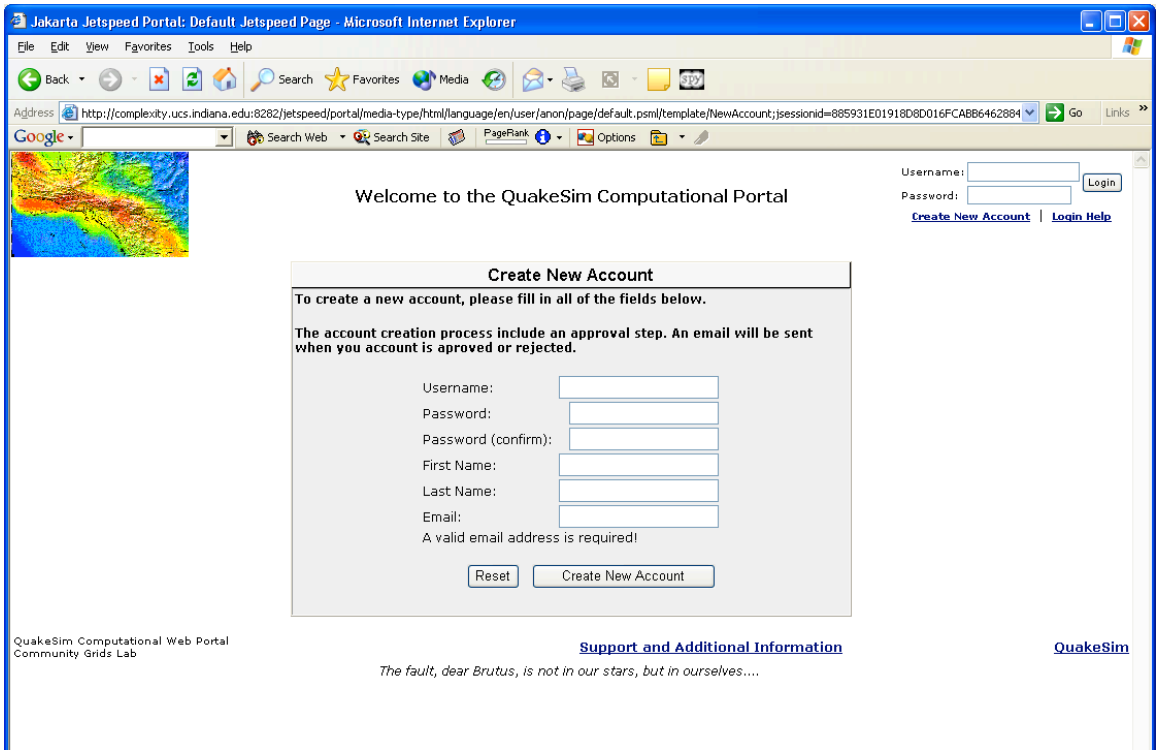

Figure 3 Creating a user account

Logging In and Customizing Your Environment

After your account has been activated, you will be notified by email and can log in using the username and password specified at account creation time. Your default portal layout will resemble the screen shown in Figure 4. The links and tab panes will be described later.

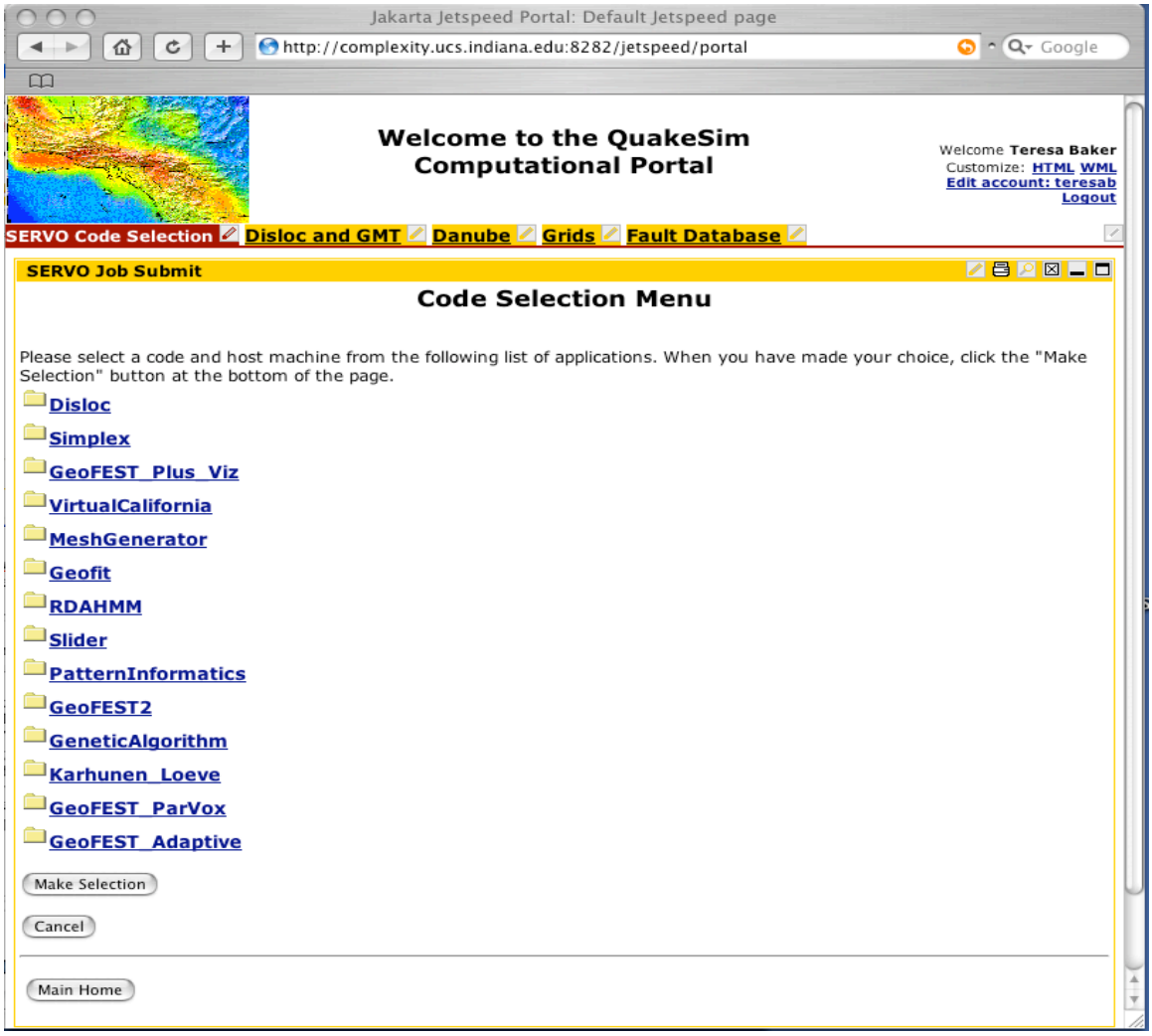

Figure 4 Default login screen

The QuakeSim portal is built using the Jetspeed portal development toolkit and provides several built in features. You may customize your colors, layout, and display tabs as desired through the customization interface. To access the customization interface, click the “Customize: HTML” link in the top left hand corner after logging in.

The initial view allows you to edit layouts and color schemes. The default layout uses tab panes. An alternative (menu pane) layout is shown in Figure 6.

You may also add (and delete) panes from your default display. From the customization view, with the default Tab Pane display, click “Add Pane’. After naming the pane (“BBC News”) click the newly created “BBC News” link. Click the now active “Add Portlet” button. You should see a page similar to Figure 7. Click the checkboxes by the desired portlets and accept all additions. You should see a new tab (BBC News) added as the rightmost tab. Clicking this link will display the recently added portlets, as shown in Figure 8. You may arrange the display in the BBC News tab as you wish through the layout customizer, described above. The display shows a two-column layout.

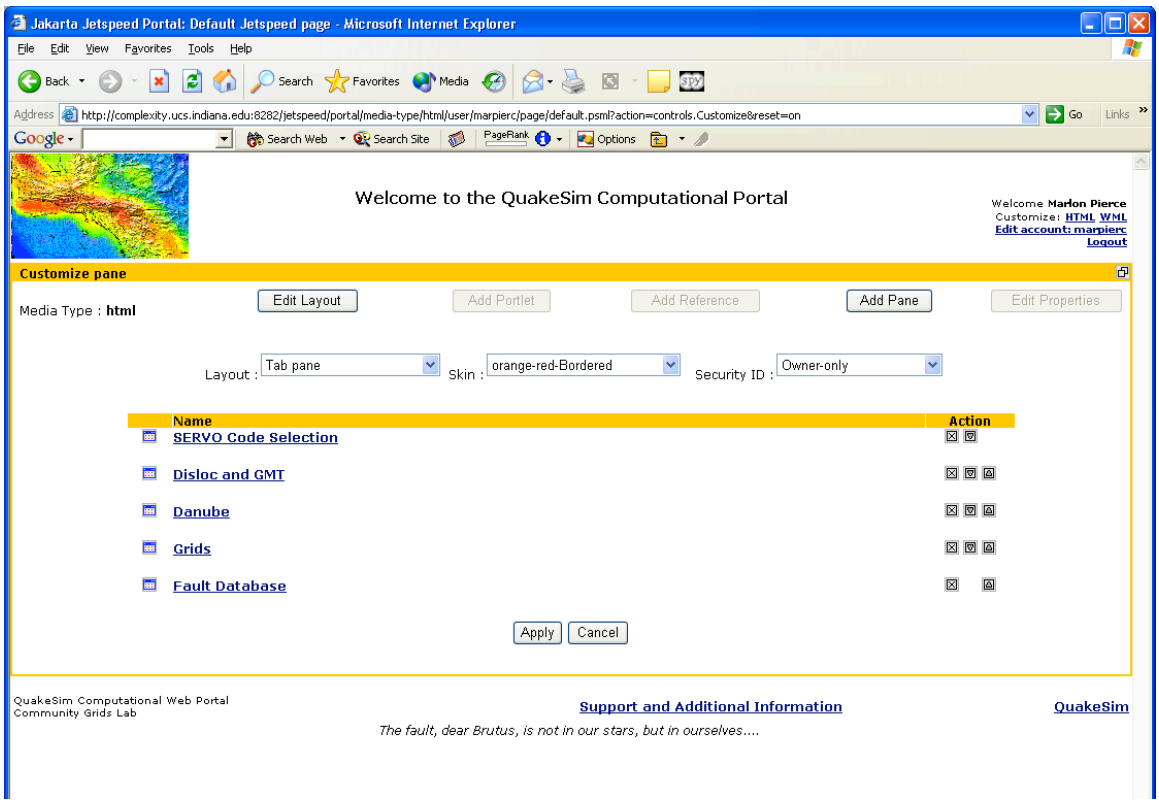

Figure 5 Cutomization main menu screen

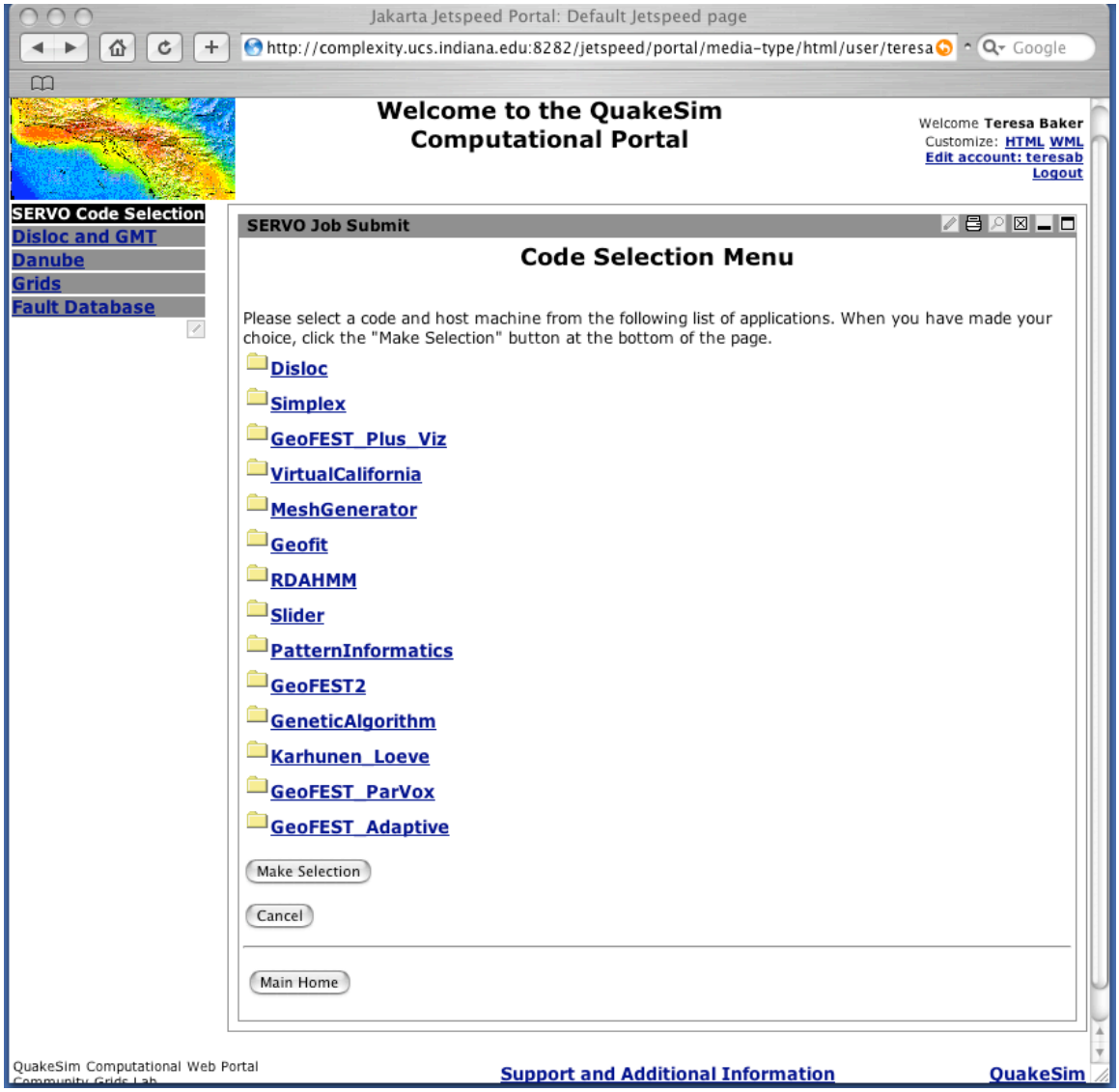

Figure 6 Menu style layout with grey borders

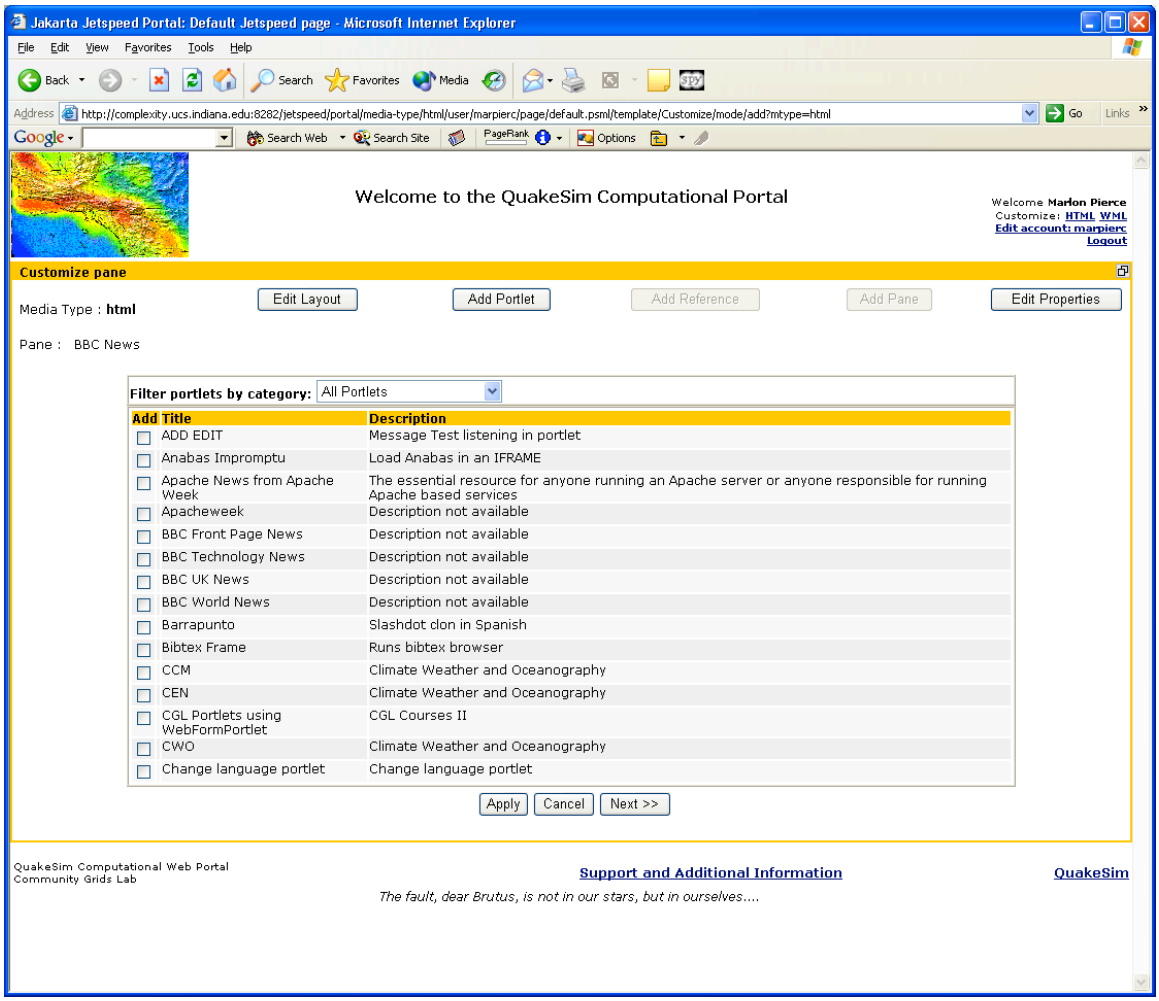

Figure 7 Portlet menu page

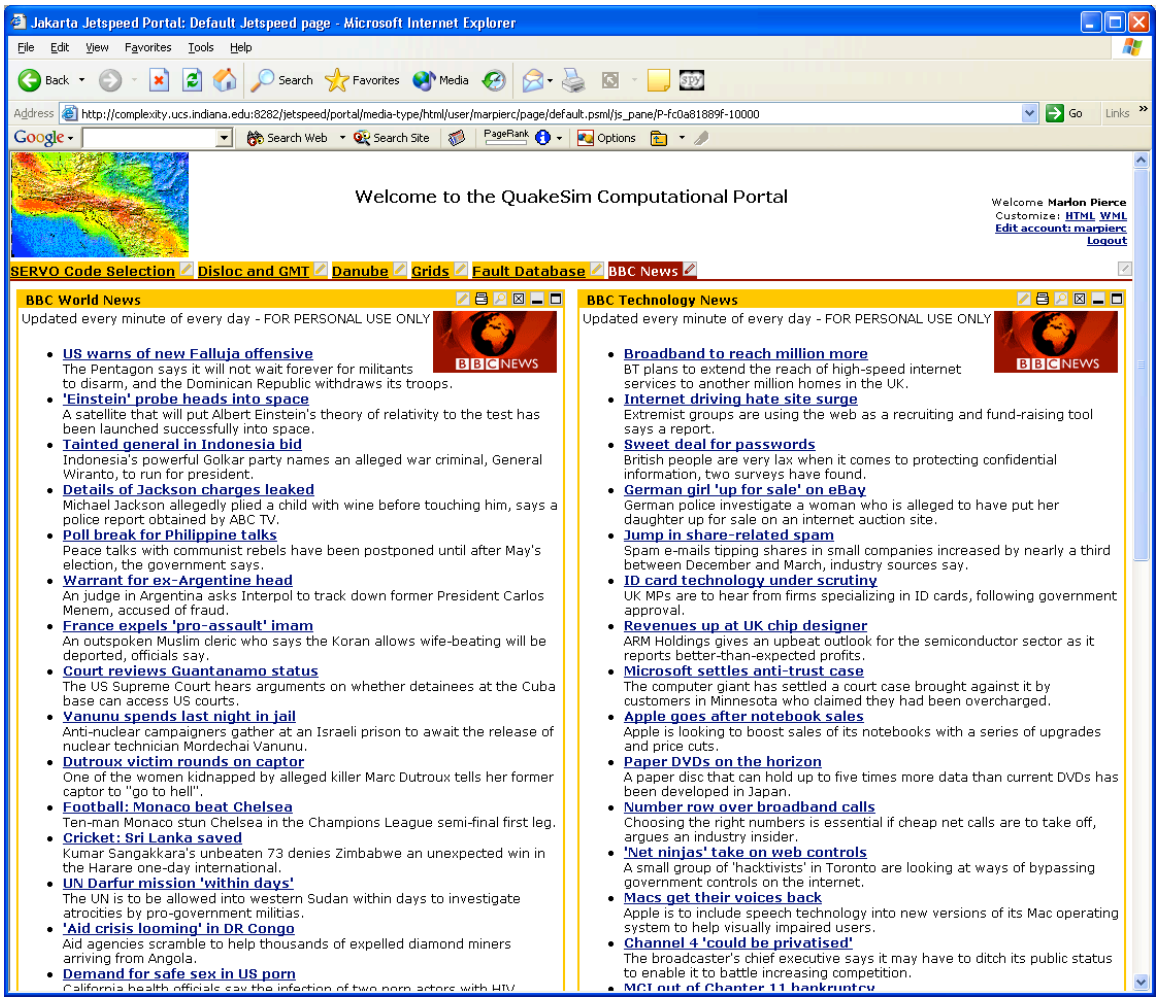

Figure 8 Newly added portlets after user customization

Running Applications

The remainder of this document shows how to run various applications in the web portal. These user interfaces follow a similar format, but readers should expect new capabilities (such as visualization of output) to be added to several of the portals.

Each of the applications is organized around the concept of project folders. When you select a particular application from the Code Selection Menu (Figure 4), you will typically have the following options:

1. **Create a new project folder**. This allows you to create a new project for that particular application. Default values are typically provided for all web forms.
2. **Load an old project folder.** This allows you to load a previously created project, with stored parameters, for resubmission.

3. **View archived data, by project**. This option allows you to download output files created by chosen application in a particular session. Some applications include an optional visualization capability here.

Some applications may have more options, as described below.

The default portal entry page (Figure 4) is the starting point for all navigations that we describe. In the descriptions below, it is assumed that you have created an account, logged in, and are starting at the "Code Selection Menu" page.

The portal should be accessed with Internet Explorer, Netscape 6 or 7, or Mozilla. ***Please do not use Netscape 4.x, which has a well-known HTML table bug that will affect the portal.***

Running Disloc with GMT Visualization

From the "Code Selection Menu", select Disloc. You will then be prompted to run the application on a selected host. Select the desired host's radio button and then click the "Make Selection" button. You should see the screen shown in Figure 9. The disloc application is described in http://www.servogrid.org/slide/GEM/Disloc/disloc.pdf.

As described under "Running Applications", you may start a new project, resubmit archived projects, or view archived data. This menu selection is depicted in Figure 9. For Disloc, the project creation step consists of two steps:

1. Setting up the surface observation points
2. Setting up the fault geometries.

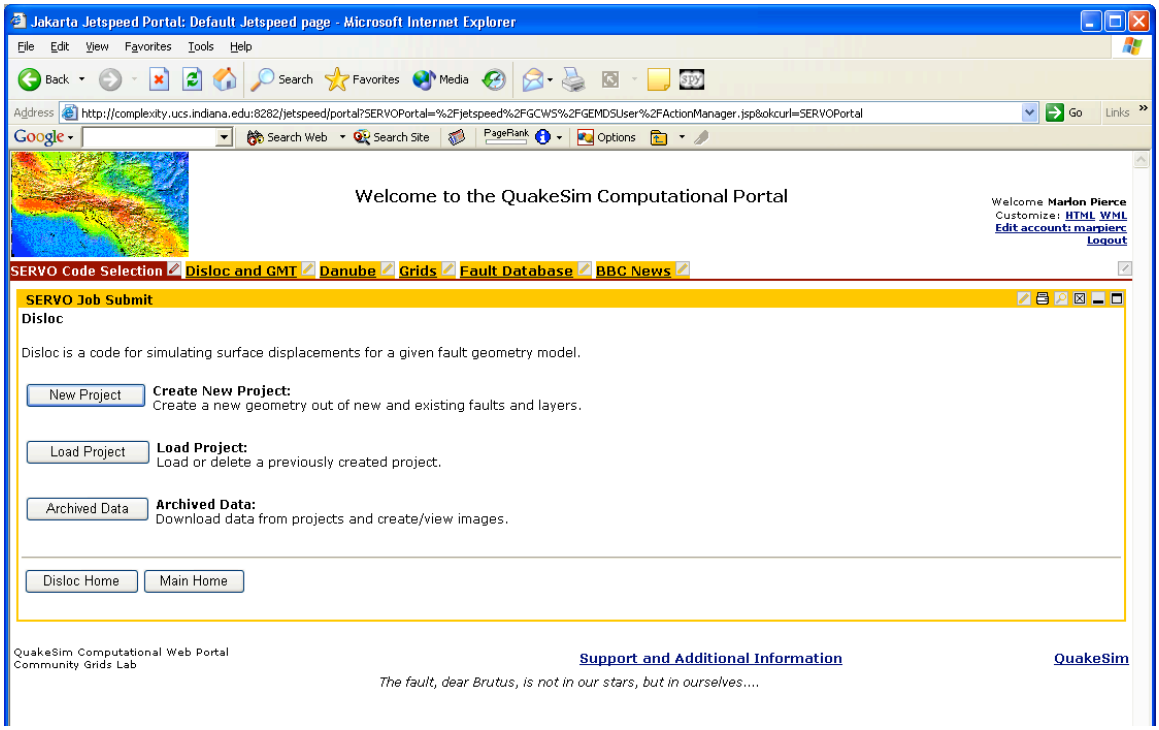

Figure 9 Disloc main navigation page

Selecting “New Project” will take you to the screen shown in Figure 10. This form allows you to specify the surface observation points (step 1 above), as well as specify the number of faults you want to use in your model.

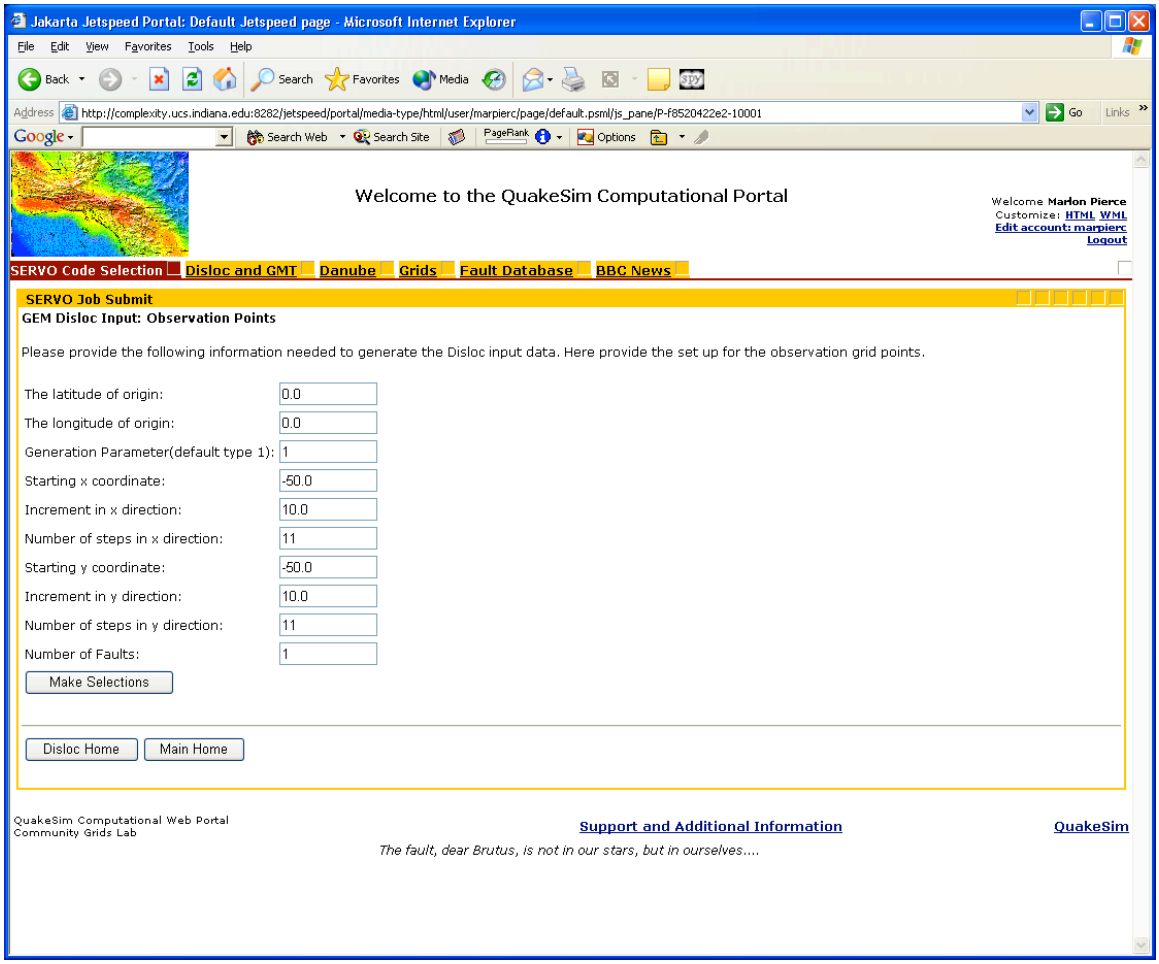

Figure 10 Disloc surface observation forms

The second step, shown in Figure 11, shows the forms for specifying the fault geometry. The figure in the screen shot shows the definitions of the fault parameters. The forms provide default parameters. Any changes made to the parameter selections will be saved in this project folder.

After specifying the fault model, you may then run the application by clicking “Make Selections”. Your results may be downloaded through the “Archived Data” track shown in Figure 9. These will be organized by date. You may also revisit previous projects by selecting the “Load Project” button from Figure 9. You will see a screen similar to Figure 12. Select the project you wish to reload. This will take you to the surface and fault parameter specification sections (Figures 10 and 11), using the previously provided parameters. During a long run, you may check the job monitor by clicking the tab for the host server.

Note your changes here will overwrite the project. It is possible to subdivide the project into subfolder sessions, but this is not currently implemented in the interface.

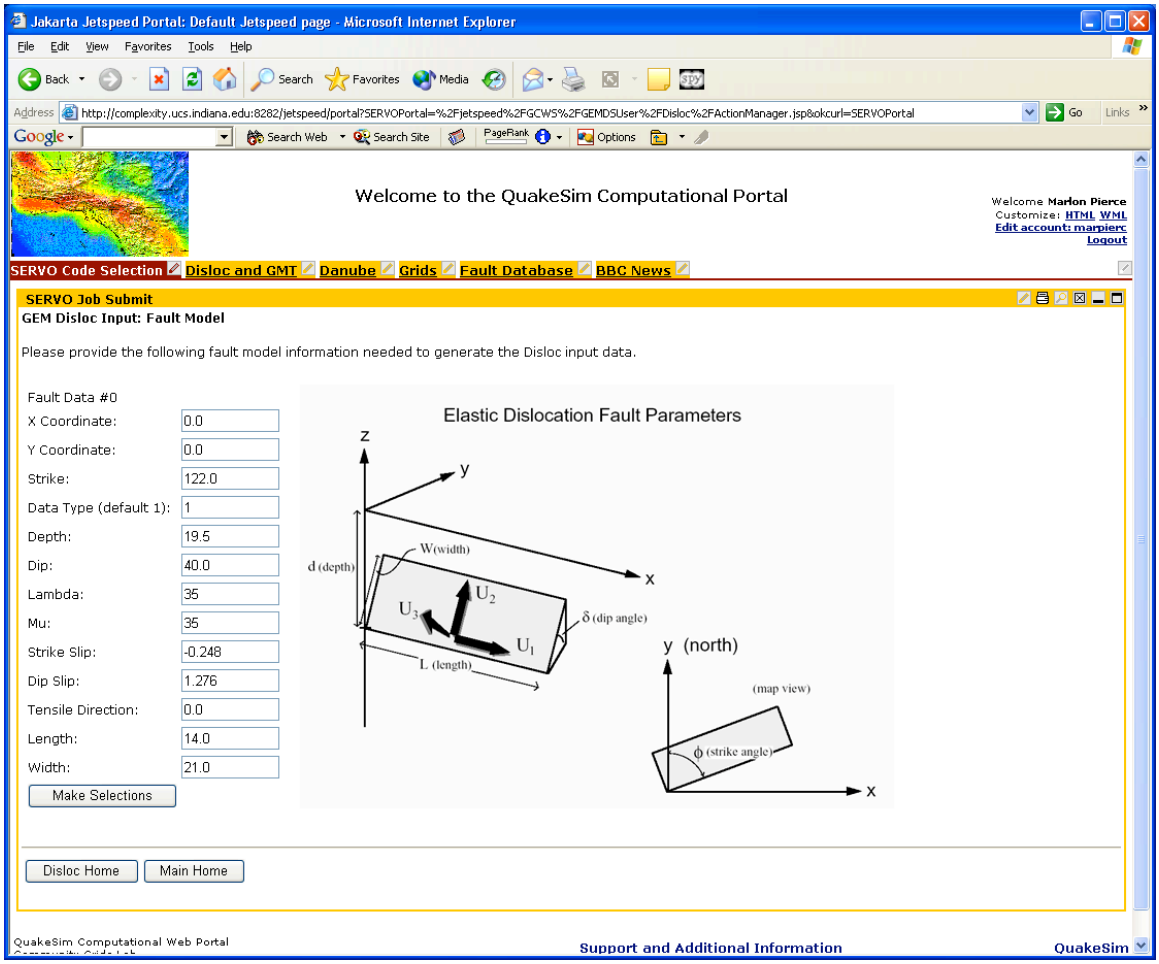

Figure 11 Fault geometry specification

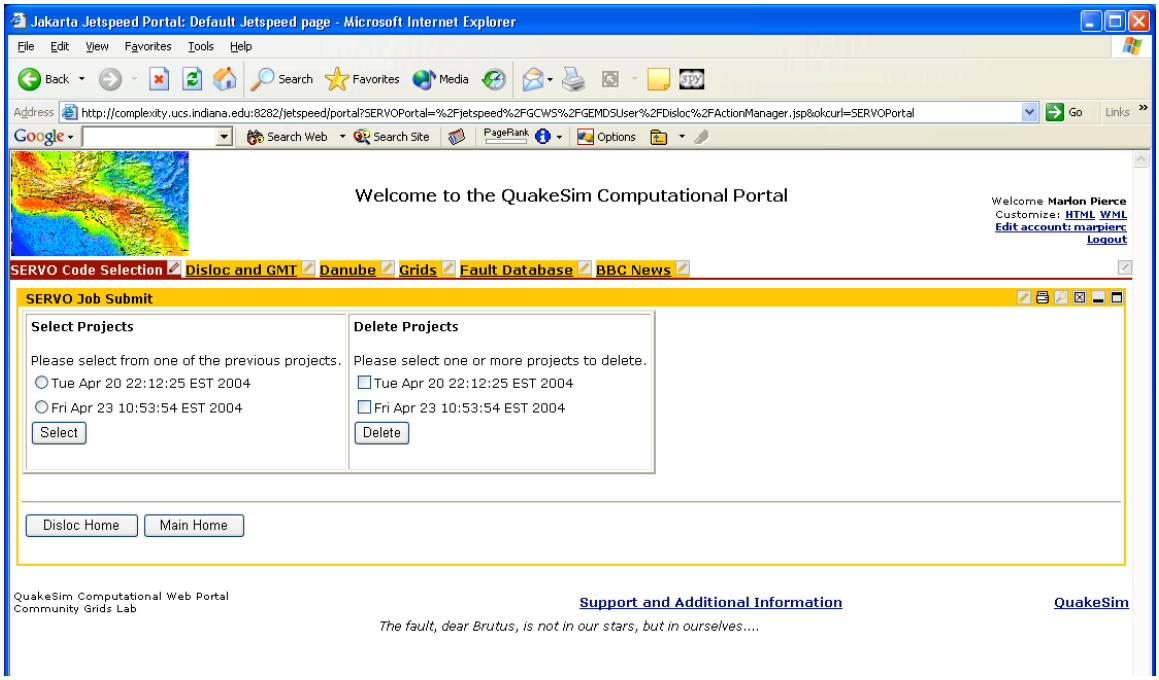

Figure 12 Archived project list.

Following a Disloc submission, you may use wrapped GMT (Generic Mapping Tool) services to visualize Disloc output. You may access archived sessions for visualization through the "Disloc and GMT" tab across the top menu bar, next to the "SERVO Code Selection" tab. After clicking this tab, you should see a screen similar to Figure 13. This list is the same as the "Archive" listing accessed through the menu, Figure 9.

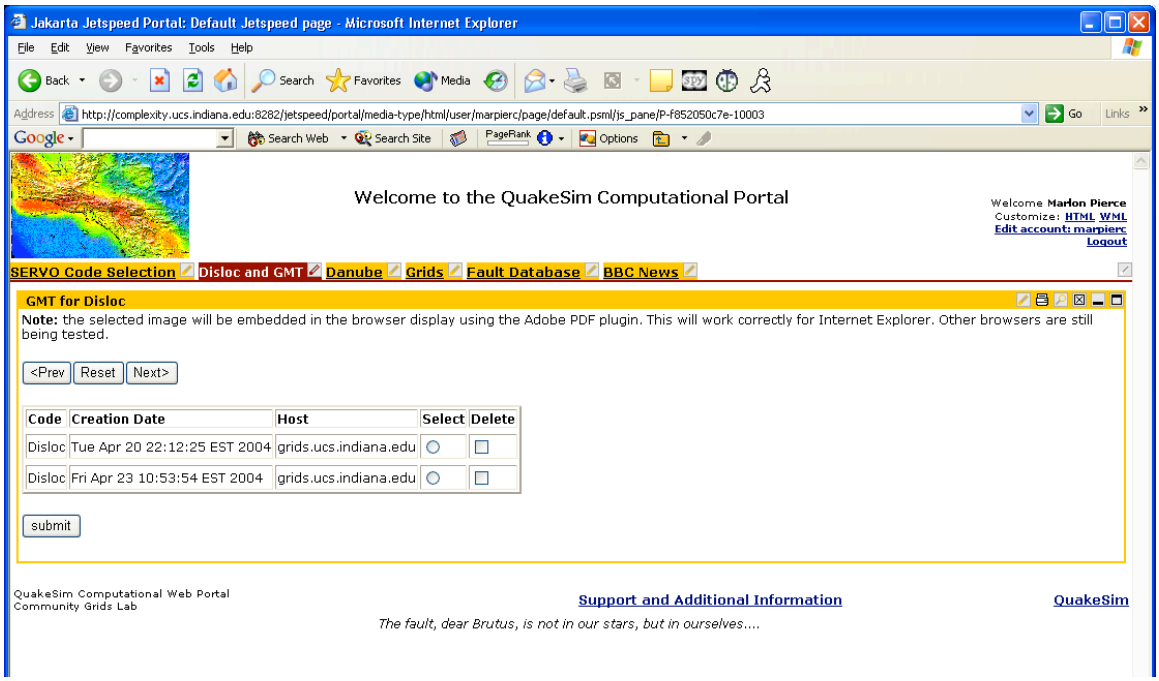

Figure 13 Session list of Disloc runs available for visualization

The image shown in Figure 14 is a sample Disloc output visualized with GMT. This figure is converted to PDF formatting for simple display embedded in browsers. You may also open the PDF image in a separate browser frame by clicking the link above the image.

The GMT visualization service also supports more interesting images. If you run Disloc with latitude and longitude values for your surface observables (first two form elements in Figure 10), the generated GMT image will be geo-located, with coastal and river boundaries shown.

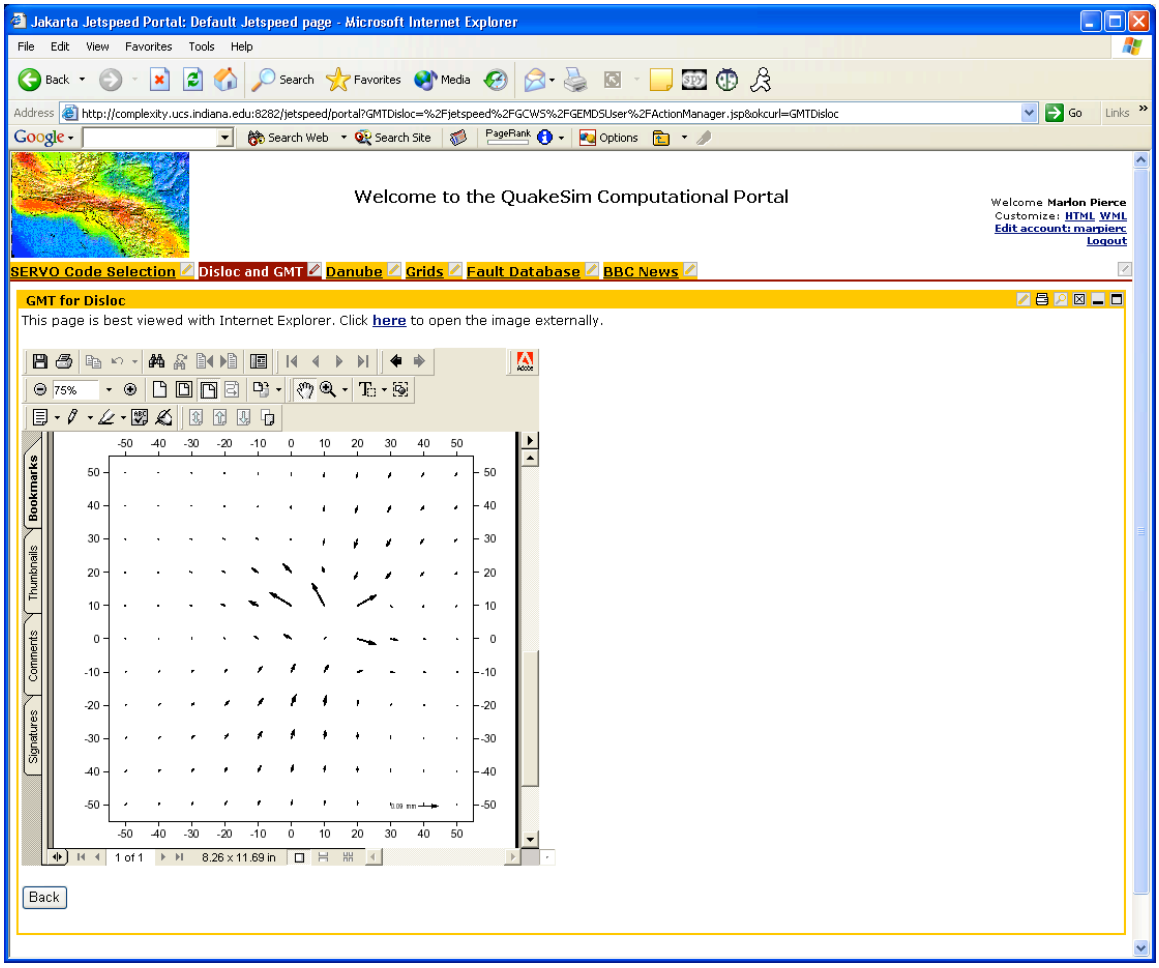

Figure 14 GMT plot of Disloc surface vectors

Running Simplex/Geofit and GMT Visualization

Simplex is a fault model estimation code: given a set of surface observations and a starting rectangular fault model (or models), Simplex finds the best fit for the fault. More information is available from
http://www.servogrid.org/slide/GEM/Simplex/simplex_readme.txt.

After choosing Simplex and the desired host, you will see a menu similar to Figure 9, which provides access to the usual capabilities defined above under “Running Applications.” We refer to this screen as the “Simplex Main Menu” below.

Starting with “New Project” you will be provided with forms to assist you with creating a Simplex input file and submitting a Simplex application. These forms request information as described in the Simplex documentation.

After running applications, you may revisit old problem folders by clicking the “Load Projects” button on the Simplex main menu. This menu button (as usual) also takes you to web forms for deleting projects.

The results of Simplex calculations may be downloaded by selecting the “Archived Data” button on the Main Menu. This will take you to a screen similar to Figure 15.

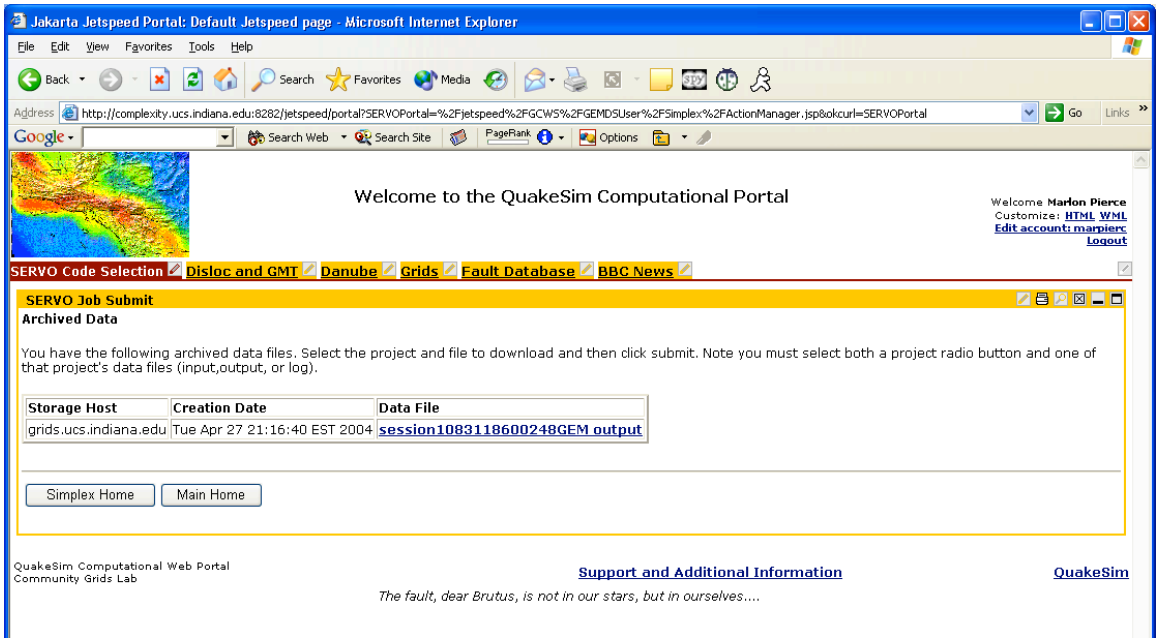

Figure 15 The Simplex data archive allows you to download results from previous simplex runs

You will see one entry in the table of Figure 15 for each project in your archive (only one in the Figure 15 screen shot). By clicking the link under “Data File” in the table, you can download the Simplex output to your desktop.

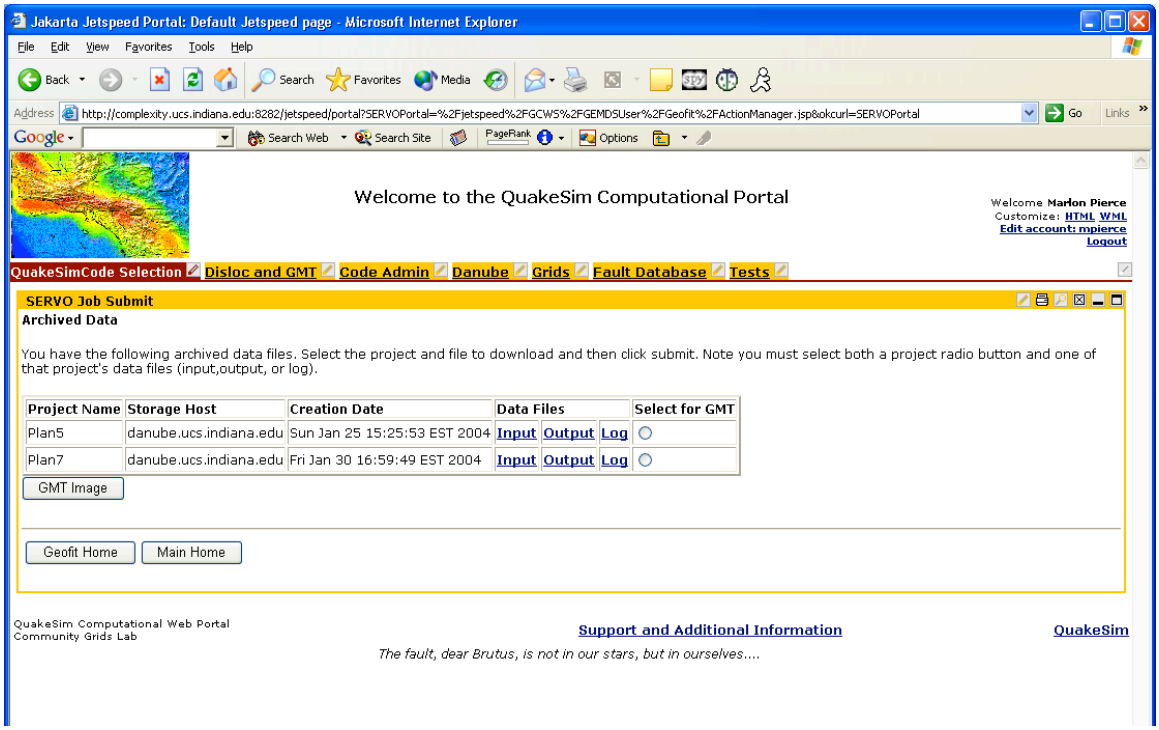

Figure 16 The Geofit data archive page allows you to download output and generate GMT images

The QuakeSim portal also provides access to a variation on the Simplex code, called Geofit. Geofit web forms may also be accessed through the code selection menu (Figure 4). Geofit output files may be downloaded in the usual way, but the Archived Data forms also provide access to GMT visualization of the output. By clicking the radio button next to the desired output data and selecting the “GMT Image” button beneath the table, you may create an image similar to Figure 16 using a wrapped GMT service.

The visualization service for Geofit converts GMT output into PDF format for simple integration into the portal display. Older browsers may have trouble displaying the embedded PDF image. The image may be downloaded from the portlet window by clicking the link immediately above the image.

The underlying GMT service has more options and capabilities for modifying the resulting image than are currently supported in the browser interface.

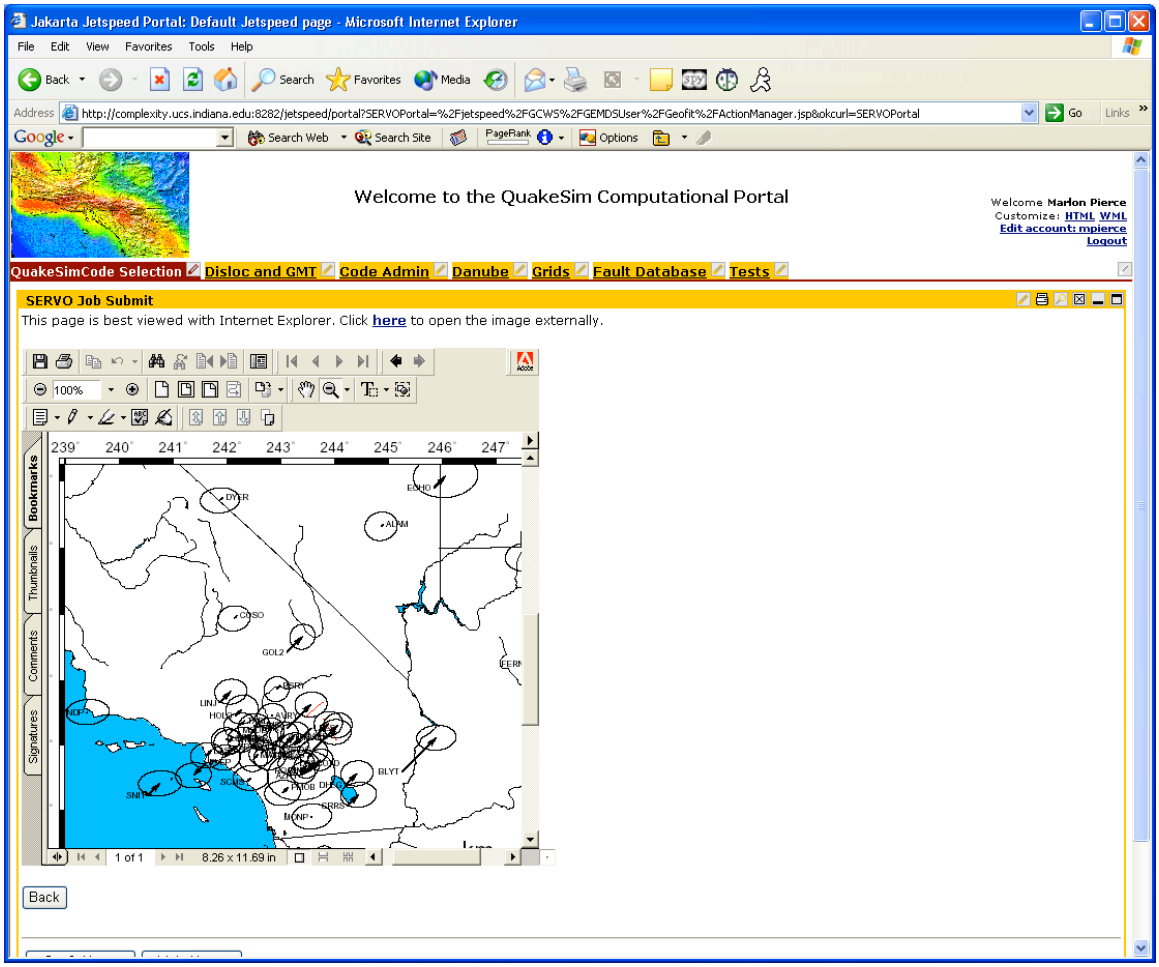

Figure 17 GMT visualization of Geofit output

Using the Mesh Generator

The QuakeSim portal includes a finite element mesh generation tool. This allows you to specify a layer and fault geometry, apply, and iteratively refine a finite element mesh. The mesh refinement is performed with the APOLLO/AKIRA tools described in the appendix.

We describe here how the Mesh Generator may be used as an independent application. It has also been integrated with the GeoFEST applications described below. It has also been integrated with the sample Finley application provided by the University of Queensland Quakes team. The Finley application is not included in the current documentation.

To use the mesh generator in standalone mode, simply click its link from the Code Selection Menu. You will see the usual application Main page, shown in Figure 18. As usual, you may start a new project, load an earlier project (or delete it), or fetch

previously generated meshes for download. The last case may be useful if you want to download the refined mesh for a desktop application or an application otherwise external to the portal.

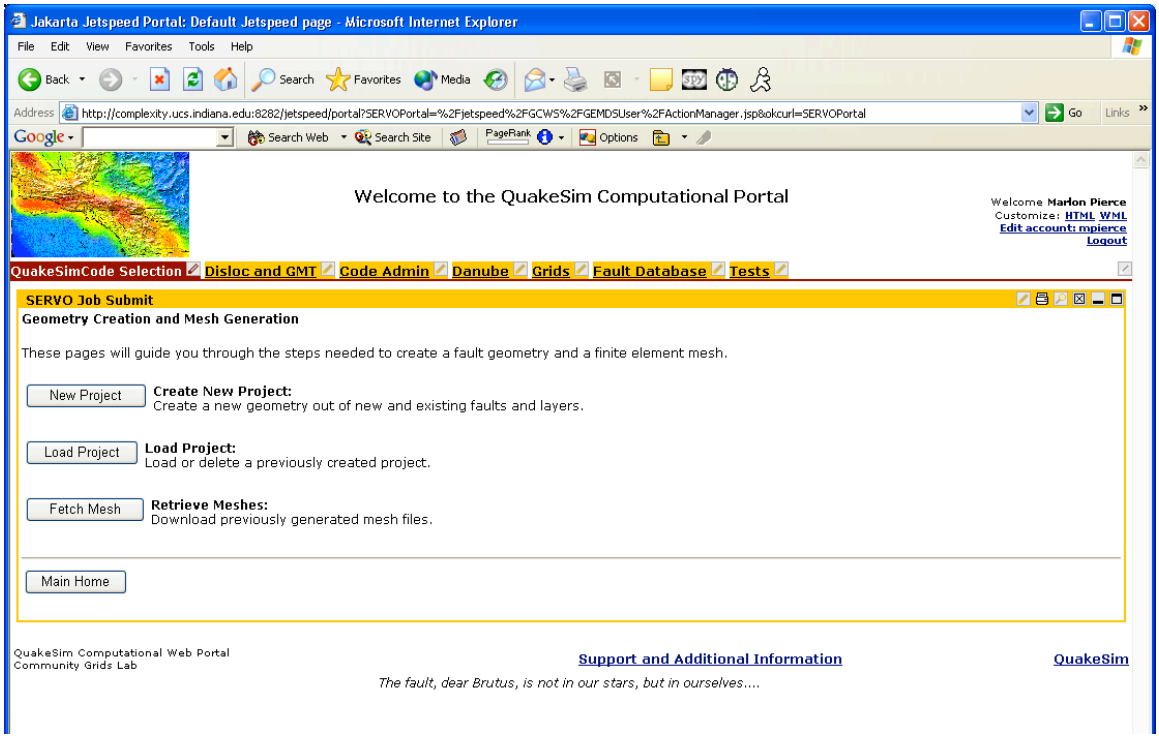

Figure 18 Mesh Generator main navigation menu

If you select “New Project” you will be prompted for a name for your application. This project will be afterwards accessible by that name. You will then see a project component selection menu similar to Figure 19. You may use this menu to add layer and fault components to your project. As described below, these components may be edited, saved, and removed at any time.

Step 1: Creating a layer geometry

From the component menu (Figure 19) you may either select faults and geometry layers stored in the QuakeSim Fault Database (“QuakeTables”) or you may create a new layer model, as prompted by the forms. As described elsewhere, QuakeTables currently contains California fault information, and not all faults include geometric models. There are also (at time of writing) a very limited number of layer models stored in QuakeTables.

You should first select the radio button “Create New Layer” and click the submit button. You will see a form menu similar to Figure 20. If you prefer, you may instead choose “Add Layer from DB” and view some predefined layer geometries. Complete the form and click “Select” beneath it. You will see your layer component added to the

component display (Figure 20). The current mesh generator only supports three layers. You will also see an option “Create Initial Mesh”, which you may choose at any time.

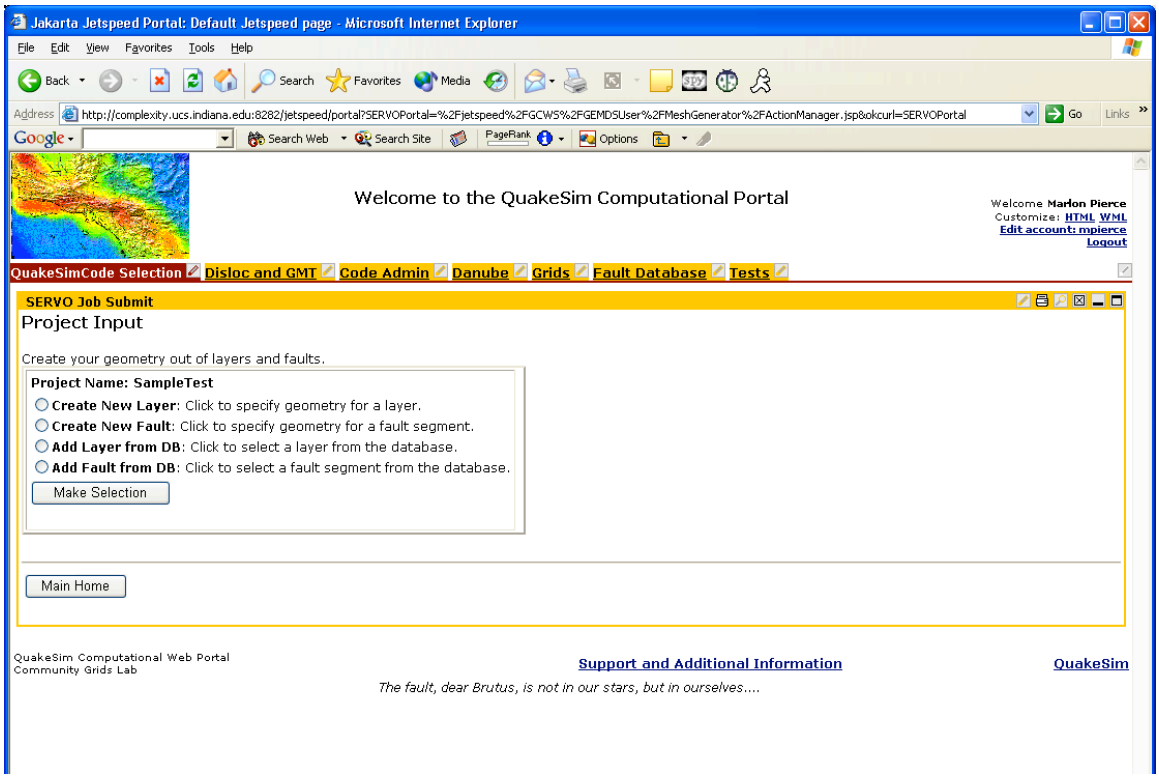

Figure 19 The layer and fault selection panel allows you to create your initial geometry

Note that you must fill out the form completely. Not all parameters (such as the material properties) are needed by the mesh generator but must be provided for specific applications (such as GeoFEST) that are coupled to the mesh generator.

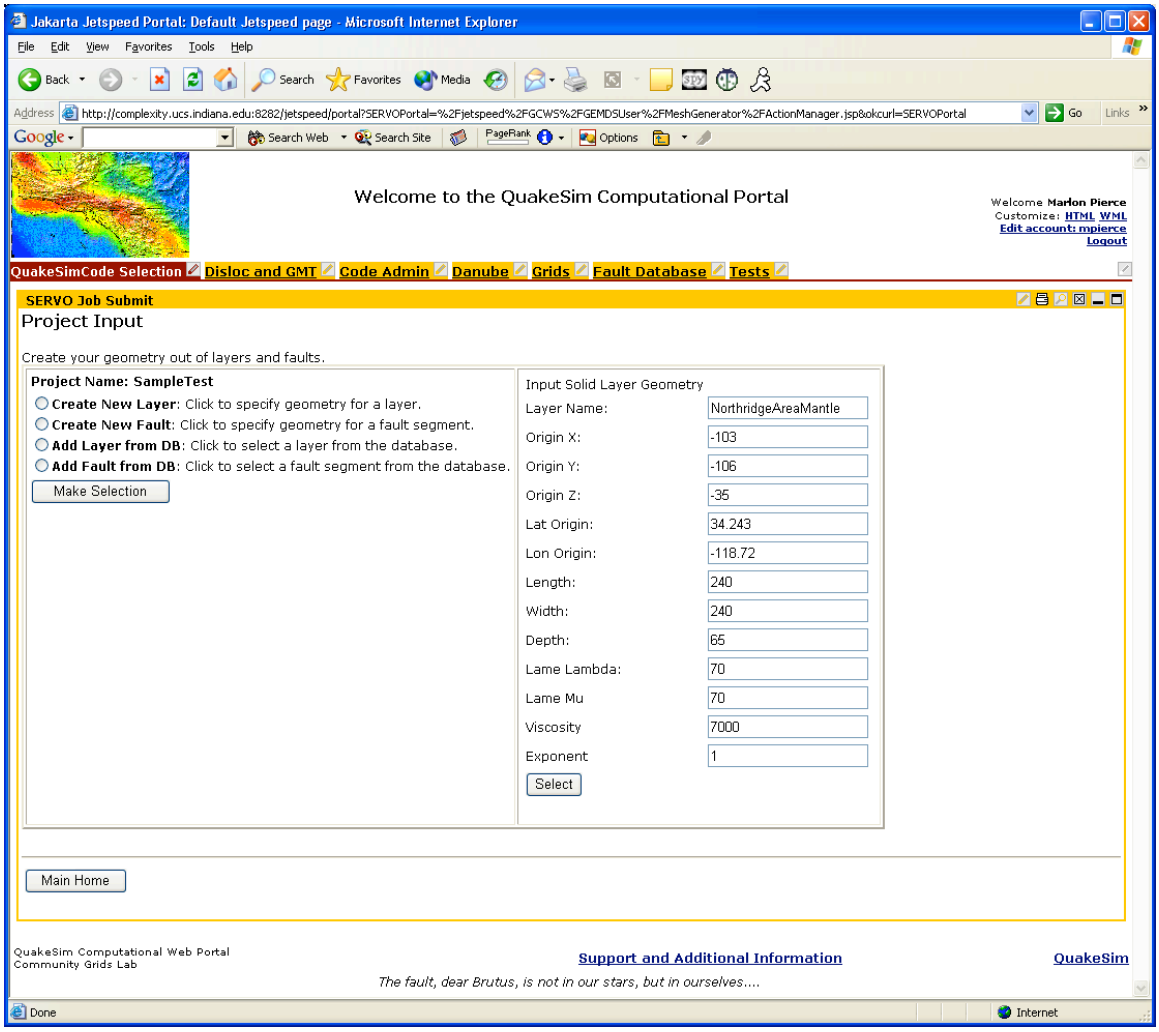

Figure 20 A sample form for describing a layer model

Note that all layer and fault components in this project will be saved and redisplayed if you reload this project later. You may log out at anytime and return to this project later.

Step 2: Add Faults

Faults may be added in a similar manner to the layer geometry definitions: you may define your own or you may select entries from the database. Note that the current mesh generator only supports one fault. When a user selects “Add Fault from DB”, the user can search the database according to Lat/Lon bounding box or Author parameters. Additionally, the user can view all faults, as in Figure 22.

Select the desired fault and click “SelectFaultDBEntry”. You will see a Web form similar to the layer specification form, Figure 20. Note that not all faults have geometric models, so you may have to supply some or all of this information in the Fault forms.

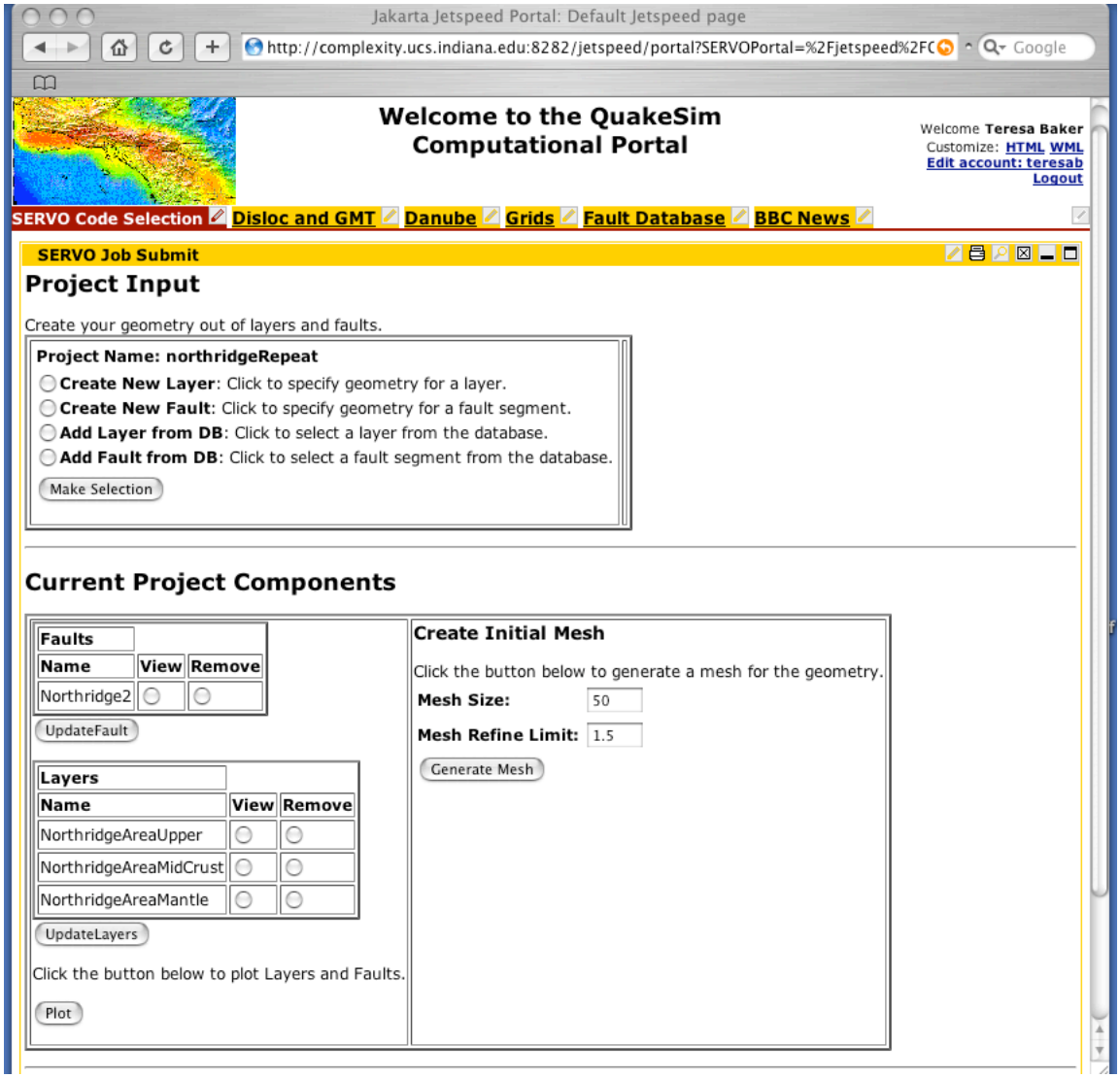

Figure 21 The project component menu displays your current layers and faults. You may edit or delete these.

Improvements on the fault selection menu will be made in the future. Possible improvements include additional filtered searches and navigable maps.

Additionally, visualization tools for fault/layer geometries have been developed and added to the portal. These are available by selecting “Plot” in the Current Project Components area.

Step 3: Generate the Mesh

After you have specified your fault/layer geometry, you may now apply mesh generator to it. Click the button “Generate Mesh” in the “Create Initial Mesh” area of the display, Figure 21. The parameters used by the mesh generator, “Mesh Size” and “Mesh Refine Limit”, are the same as used by the standalone mesh generator tool described in the Appendix. Please refer to that documentation for more information.

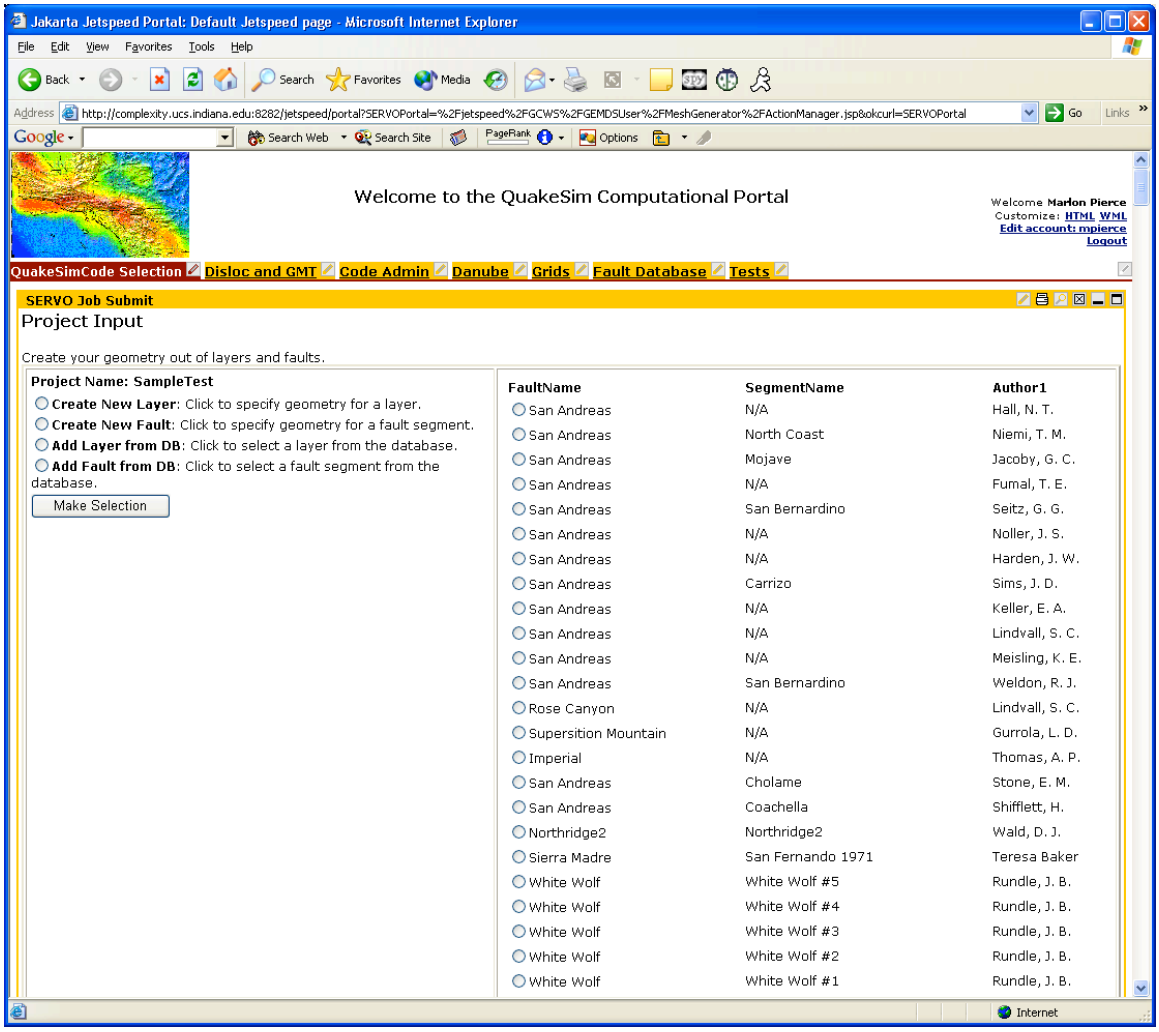

Figure 22 Select the desired fault from the fault database

The mesh generation screen is shown in Figure 23. You should iteratively refine the mesh by clicking the “Refine Mesh” button until the mesh has sufficiently converged. Convergence may be determined by examining the generator’s feedback for “Number of elts with substantial priority”, the eighth line of mesh generator output in Figure 23. This number will increase monotonically until the mesh is refined; it will then begin to drop rapidly.

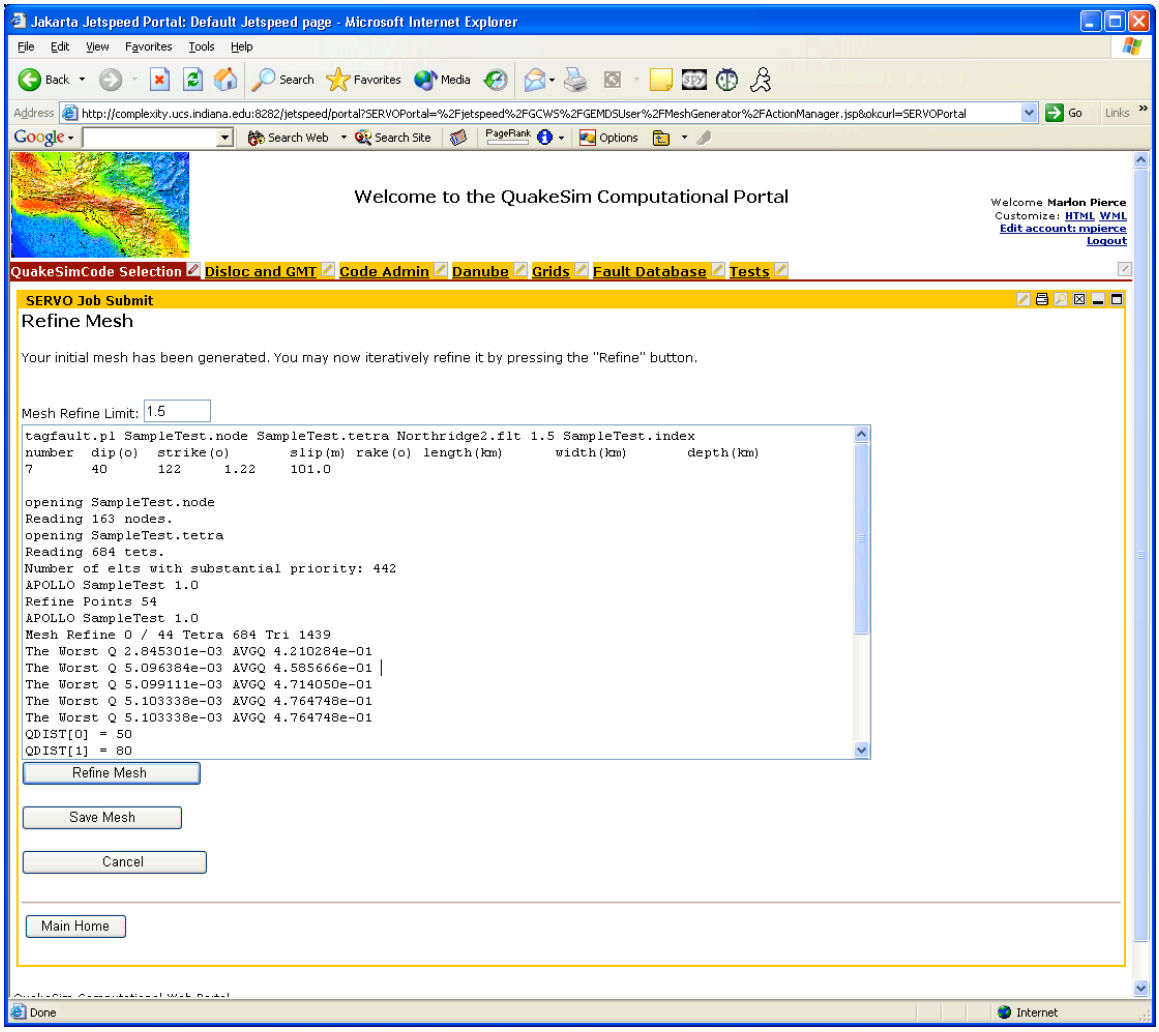

Figure 23 Mesh refiner feedback

After you have sufficiently refined the mesh, click “Save Mesh” to return to the Mesh Generator main menu. You may download the mesh by clicking the “Fetch Mesh” button.

Visualization tools for the mesh generator have been developed that will allow you to display the surface mesh for your problem. This has not been added to the portal (as of writing) but may be available.

Step 4: Downloading the refined mesh

You may download the mesh results at any time. Figure 24 shows a sample page. Each project has three files: mesh, tetra, and index. Please refer to the Mesh Generator documentation (in the Appendix) for more information. By clicking the links you may download the file into a new browser window. You may then save this to your local file system by using your browser’s “Save” feature. On Internet Explorer, this is “File->Save as…”.

Known Issues

1. Currently only one mesh set is saved per project. If you reload an old project and start refining the mesh again (that is, you reach the screen shown Figure 23), you will overwrite the previous mesh. It is possible to save more than one mesh set per project, but this is not implemented in the current portal interface.
2. The mesh refinement tools do not validate the user's geometry. The mesh generator can fail if your fault extends across a layer boundary or does not fall within the layer geometry you have specified. It is hoped that visualization tools under development will help prevent this problem.

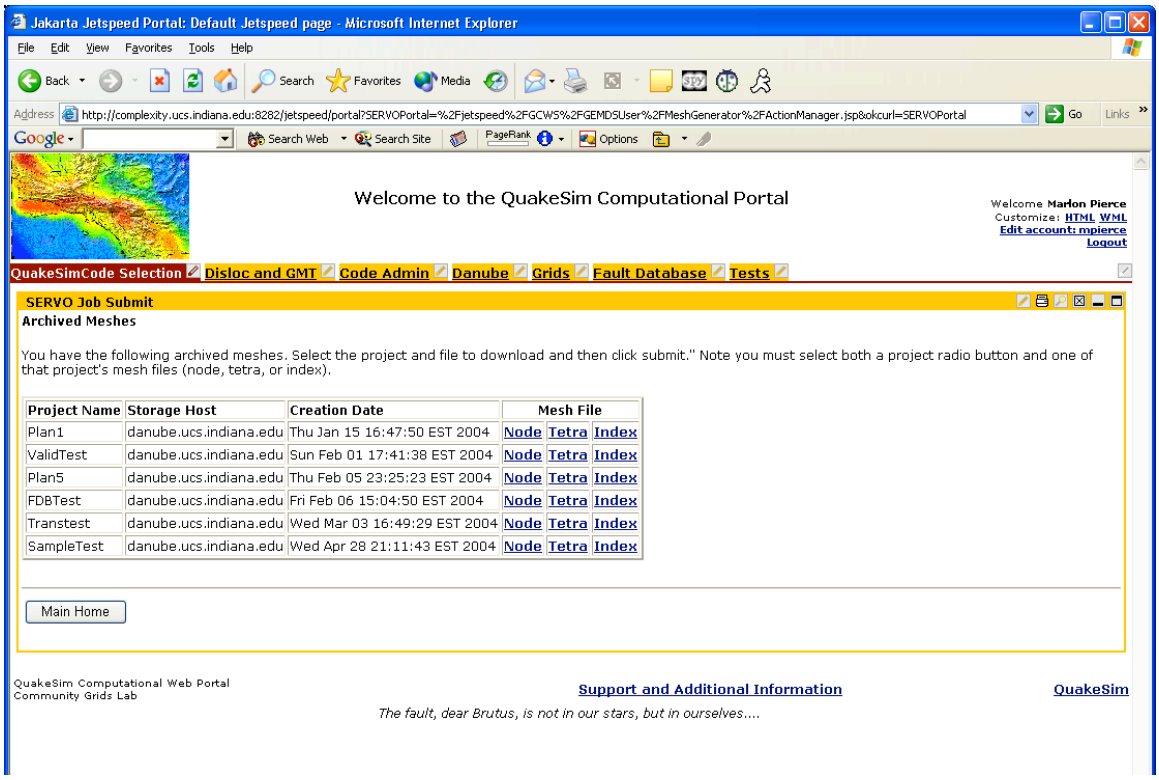

Figure 24 Downloadable meshes from previous projects

Introduction to Running GeoFEST in the Portal

QuakeSim comes with three portal interfaces to GeoFEST. The primary interface is called simply "GeoFEST2." Two other interfaces, "GeoFEST_Plus_Viz" and "GeoFEST_ParVox" are also provided. These are more limited demos that couple GeoFEST runs to RIVA and ParVox visualization tools, respectively.

- GeoFEST2 is intended for normal usage.
- GeoFEST_Plus_Viz is limited to simulating the Northridge simulation and also has a limited interface for specifying GeoFEST input.

- GeoFEST_ParVox requires that the user have access to a ParVox client program. It has a more general interface for specifying GeoFEST input parameters, similar to that used by GeoFEST2, except the ParVox simulation requires certain GeoFEST options to have fixed values. This demo also assumes that GeoFEST and ParVox services are co-located on the same host (orion.jpl.nasa.gov in the demo) because of extremely large GeoFEST output files visualized by ParVox.

Running GeoFEST

Log in to the portal as normal and select "GeoFEST2" from the Code Selection Menu (Figure 4). GeoFEST shares project folders with the Mesh Generator, so you will see the Mesh Generator menu page, Figure 18.

Before running GeoFEST, you must first specify a project layer/fault geometry and generate a mesh, as described in the section "Using the Mesh Generator." Please refer to those instructions.

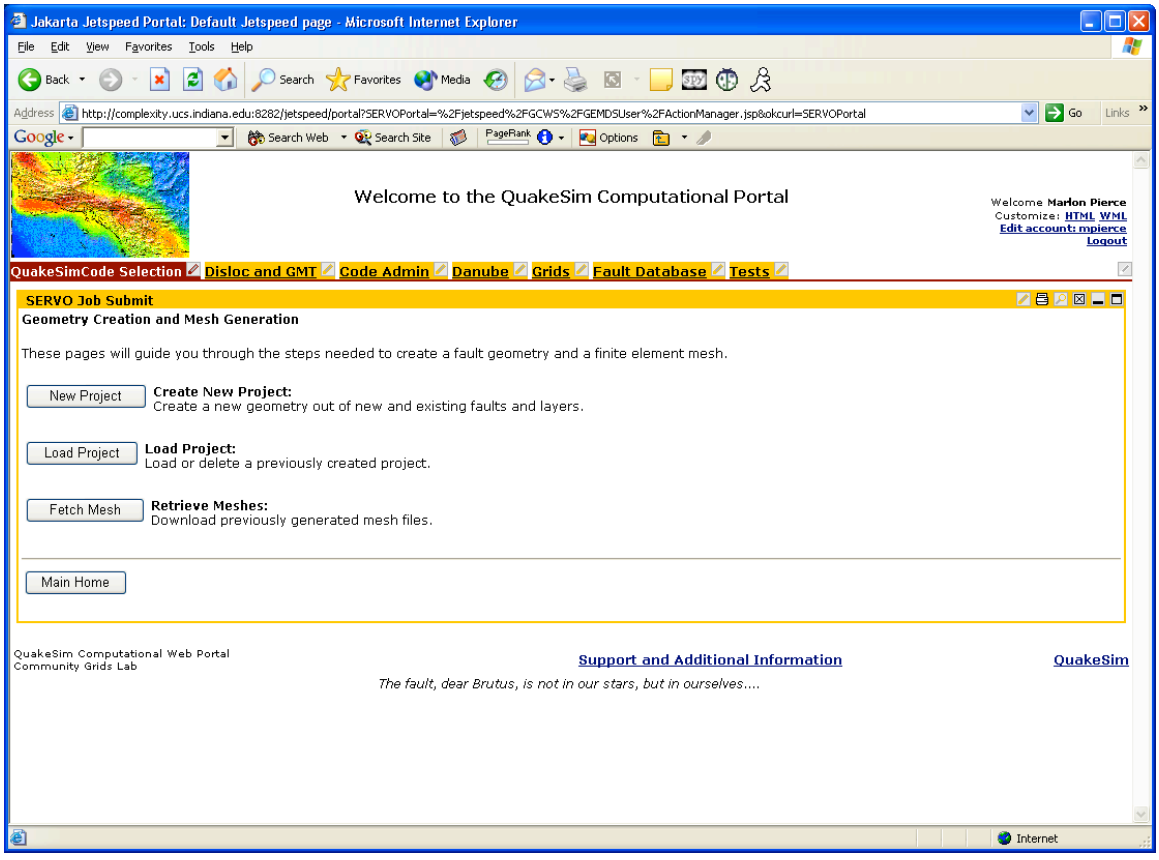

Figure 25 GeoFEST Navigation Page

After creating your project mesh, you will then be able to launch GeoFEST. Clicking the "Save Mesh" button in Figure 23, you will next fill out a form, Figure 26, to specify GeoFEST input parameters. These input parameters, the finite element mesh, and the

material properties provided in the layer definitions will be used to construct the GeoFEST input file. For detailed descriptions on these form options and information about GeoFEST in general, see http://www.openchannelfoundation.org/projects/GeoFEST/. Sample values are provided for all text fields, but you may change these as desired.

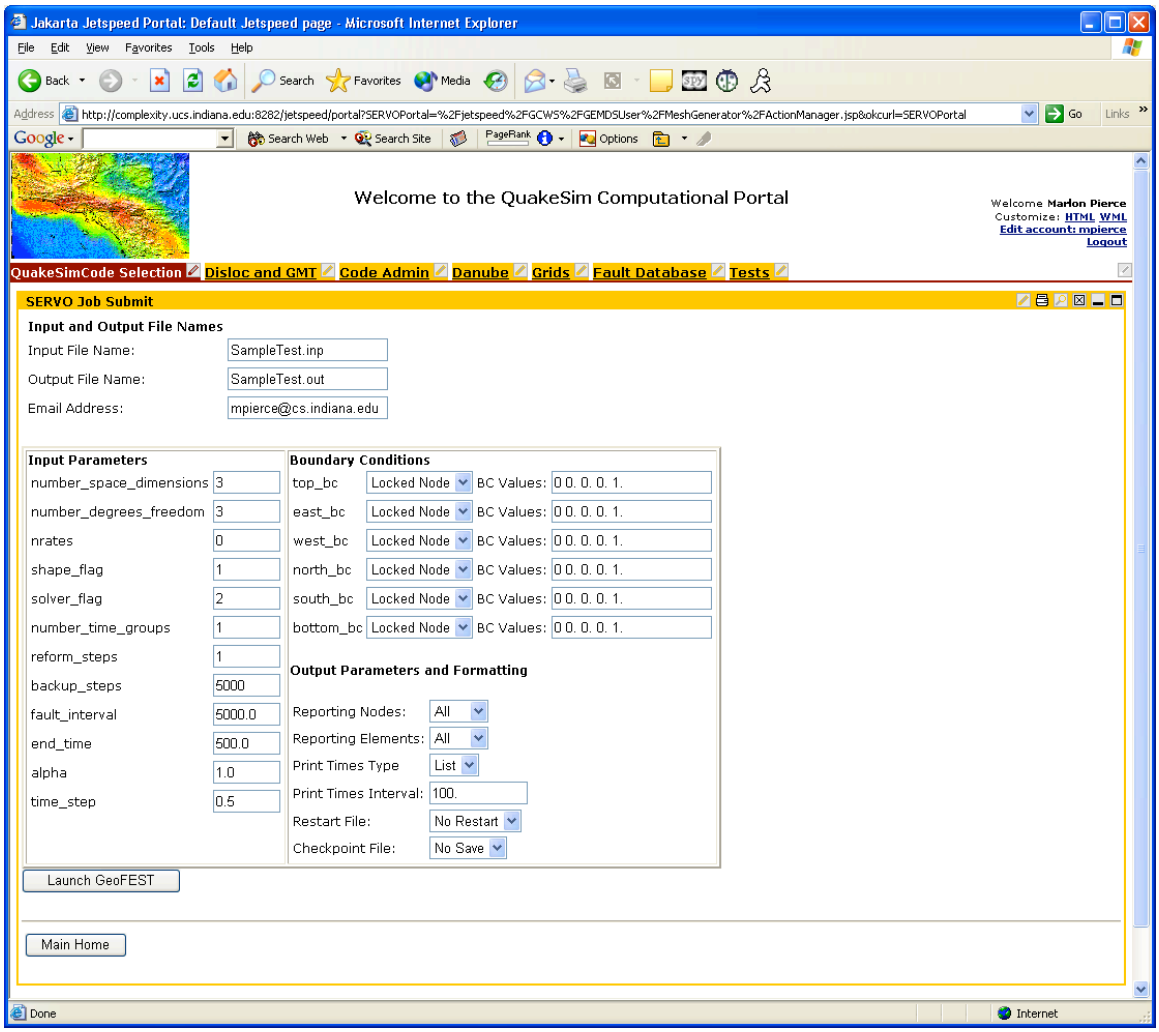

Figure 26 GeoFEST input parameter forms

Clicking this button launches GeoFEST. You may monitor GeoFEST's progress by selecting the tab across the top bar that corresponds to the selected host. For the test bed deployment, this is "Danube". Click this tab (middle top of Figure 27) to see a summary of running processes. The code will be running under the username "gateway" in the demo.

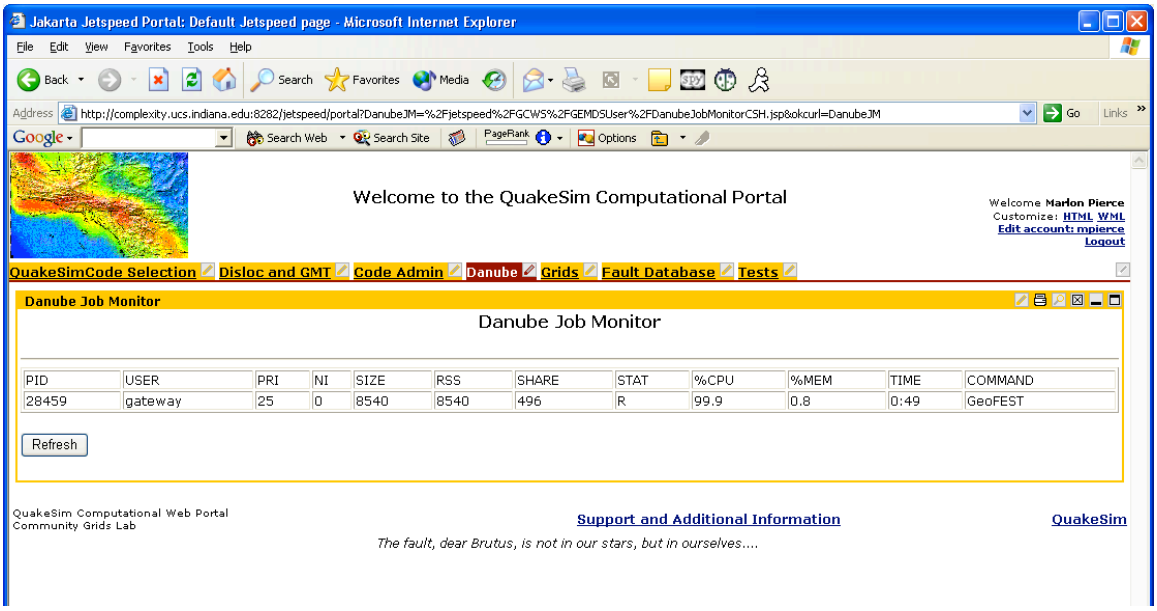

Figure 27 The job monitor for Danube verifies that GeoFEST has been launched

Completed GeoFEST data runs are available for download from the "Archived Data" section of the GeoFEST main menu. Additionally, the log file produced by GeoFEST, and the GeoFEST input file are available. Click the link next to the desired file.

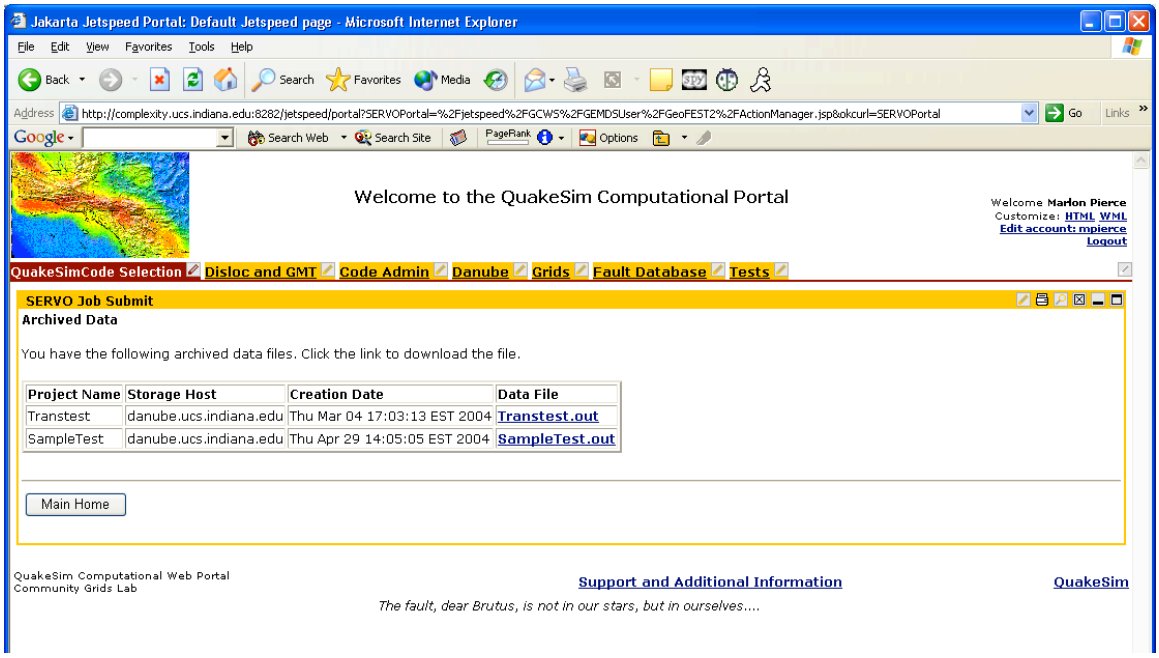

Figure 28 The GeoFEST data archive lets you download GeoFEST output

Running GeoFEST with RIVA Visualization

This particular version of GeoFEST is part of the original demonstration milestone. It may still be used to run GeoFEST and generate RIVA visualizations, but this particular track should only be used as a demonstration of the Northridge, California fault. The user interface has not been upgraded to include GeoFEST input parameter specifications (Figure 26).

Select and Setup GeoFEST

Choose the "SERVO Code Selection" tab. You should see the screen in Figure 4. Choose "GeoFEST_Plus_Viz". You will first go through the Mesh Generation phase, as described above. You should select the Northridge layers and Northridge2 fault.

Important Note: the current version of the system requires that layers be added from bottom to top. For example, the Northridge layers in the layer database should be entered in the following order: NorthridgeAreaMantle, NorthridgeAreaMidCrust, and NorthridgeAreaUpper.

Successively refine your mesh by selecting the "Refine Mesh" button. The portal will block for increasingly longer times while the mesh generator is working (a few seconds at most). As a rule of thumb, the mesh should be refined until the "number of elements with substantial priority", after steadily increasing with each refinement, begins to drop. For the sample Northridge case, this occurs when the total number of points is approximately 10,000 and the number of tetrahedrons is about 50,000.

Submit GeoFEST

After you have satisfactorily refined your mesh, you may convert this into a GeoFEST input file and run GeoFEST. You should provide values for the indicated fields or use the defaults. Enter your email address and you will be emailed a link to the movie when the job has completed.

A GeoFEST run on Danube with 10,000 nodes/50,000 elements for the Northridge simulation takes about 45 minutes. Generating the movie of the output requires another 15-20 minutes. You will be notified by email when your job has completed and your movie is available for download.

Running GeoFEST with ParVox

We have set up a version of GeoFEST to run with the ParVox visualization application. This version of the GeoFEST application assumes that it shares a file space with the ParVox server. Typical GeoFEST outputs for visualizations can be dozens of GBs, so running GeoFEST and ParVox on the same file space simplifies the demonstration.

To run this demo, you must have the following:

1. An account on orion.jpl.nasa.gov

2. Access to a ParVox client application.

Please contact Peggy Li for ParVox client applications. For accounts on orion, please contact Cris Windoffer (Cris.Windoffer@jpl.nasa.gov).

The initial GeoFEST setup procedure is identical to that described in the "Using the Mesh Generator" and "Running GeoFEST" sections. After clicking the "Save Mesh" button (see Figure 23), you will be redirected to the GeoFEST input form page, shown in Figure 30. This is a slightly modified version of the form shown in Figure 26:

1. You access orion via ssh with your Unix username and password.
2. The "Reporting Elements" and "Reporting Nodes" options are set to "All", as assumed by ParVox.

For simplicity, you may leave all parameters in their default settings. *Make sure the email address is correct.* You must provide the correct username and password to run GeoFEST on orion.jpl.nasa.gov. You may wish to change the "end_time" parameter to run longer or shorter simulations. You can also change the "Print Time Interval" value to print more or less GeoFEST output for animation. The default settings are 500 time steps with print intervals every 100 steps. You may want to change the boundary conditions from "Locked Node" to "Free Node" for more interesting simulations.

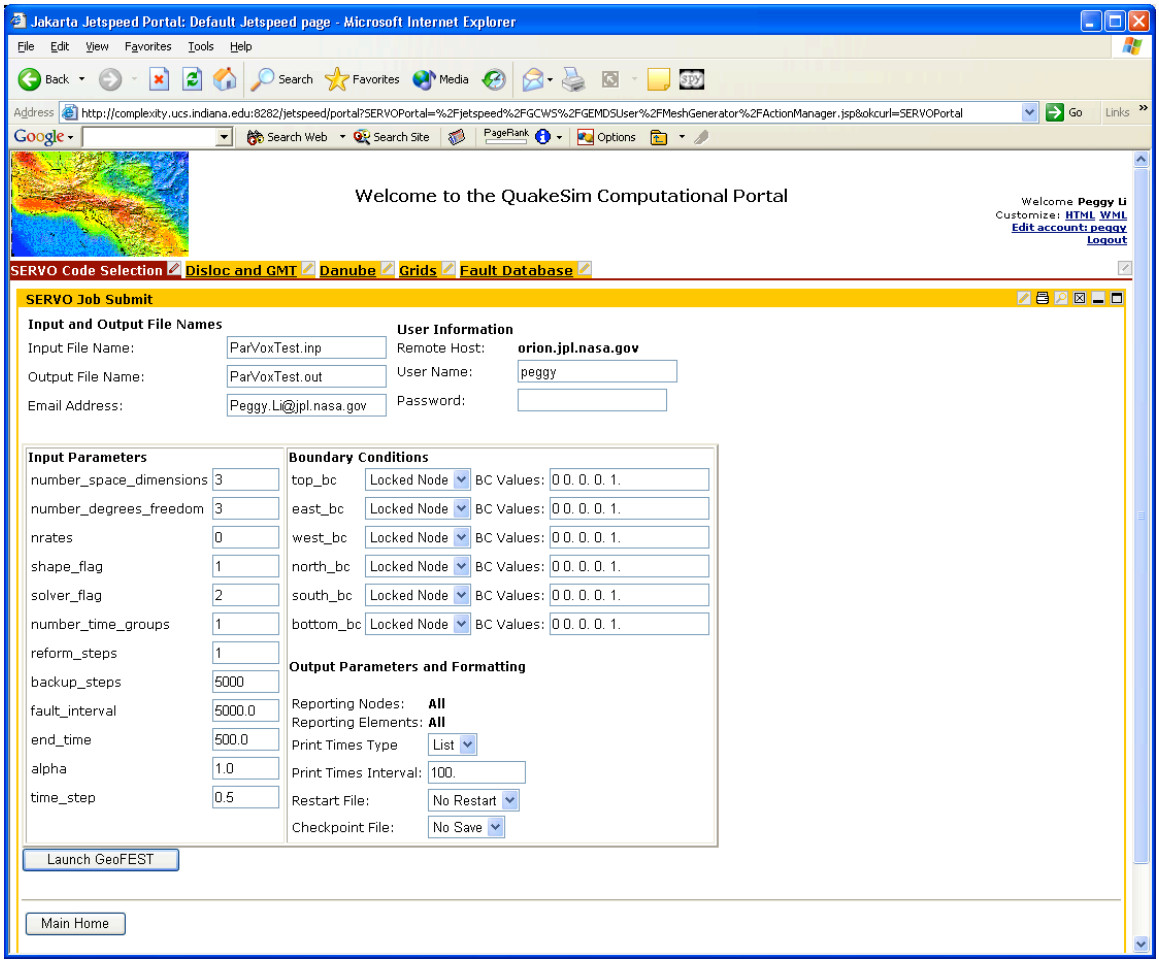

Figure 29 GeoFEST input parameters for ParVox

At writing, there is no PBS monitor portlet for Orion, so you can verify that your job launched correctly by logging into Orion and using qjobs. If you entered the "Email Address" field correctly, you will get email notifications when the job begins to execute on the Orion PBS queue.

Also, there is currently no check on the correct submission to Orion, so if you mistype your password, you will not be notified. Both of these shortcomings will be remedied.

After GeoFEST has completed, you will be notified via email that ParVox is ready for remote rendering. Log into a ParVox client and launch the command as instructed by your email message. Typically, the message will look like this.

xmparvox -server orion.jpl.nasa.gov -port 7000

Running RDAHMM

RDAHMM is a Hidden Markov Model application for data mining sequential data sets such as may be obtained from GPS and Seismic Catalogs. For general information on HMM methods, see http://www.servogrid.org/slide/GEM/RDAHMM/HMM_Tutorial.pdf.

The RDAHMM service in the QuakeSim portal may be either applied to uploaded files (suggested for testing and validation) or you may first select entries from the QuakeSim GPS and Seismic Database catalogs. See Figure 31. Select "New Project" to apply RDAHMM to an uploaded file. Select "DB Project" to first pull data from the appropriate database.

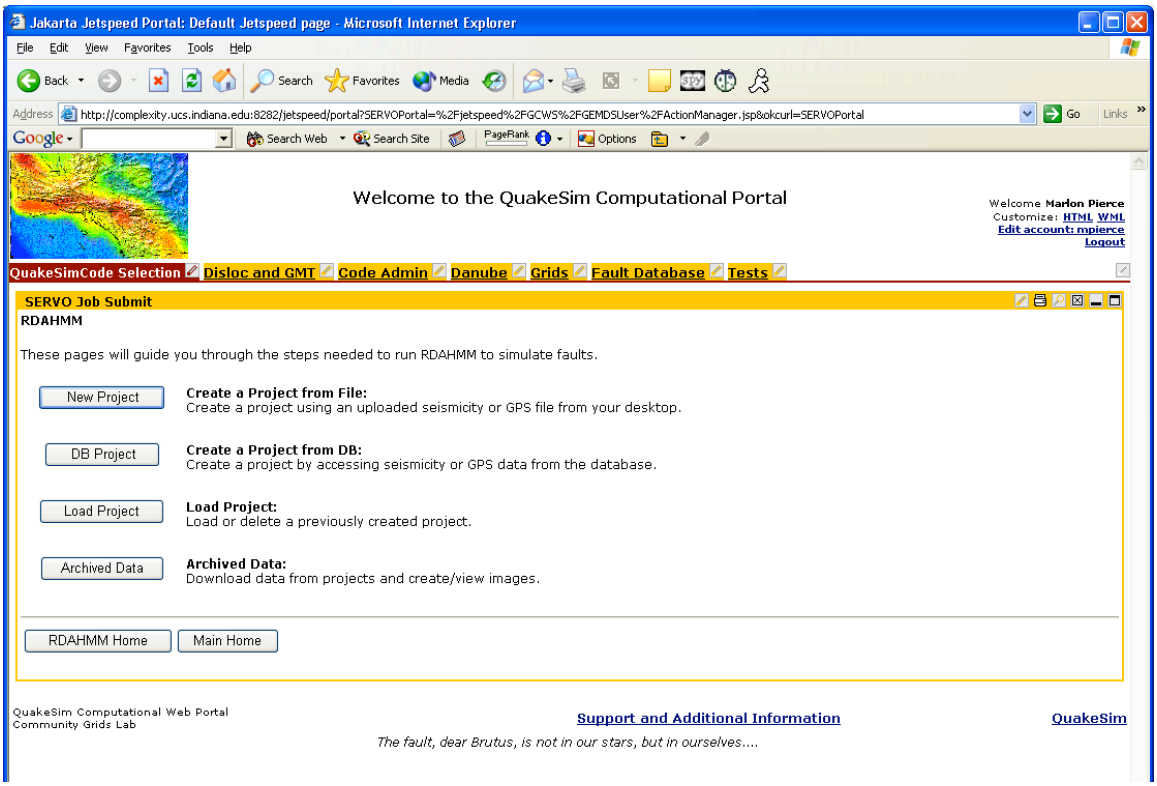

Figure 30 RDAHMM main entry page

When choosing the "New Project" option, you will see a screen similar to Figure 32. You may follow the provided links to download sample/validation test cases for RDAHMM and upload them into the portal.

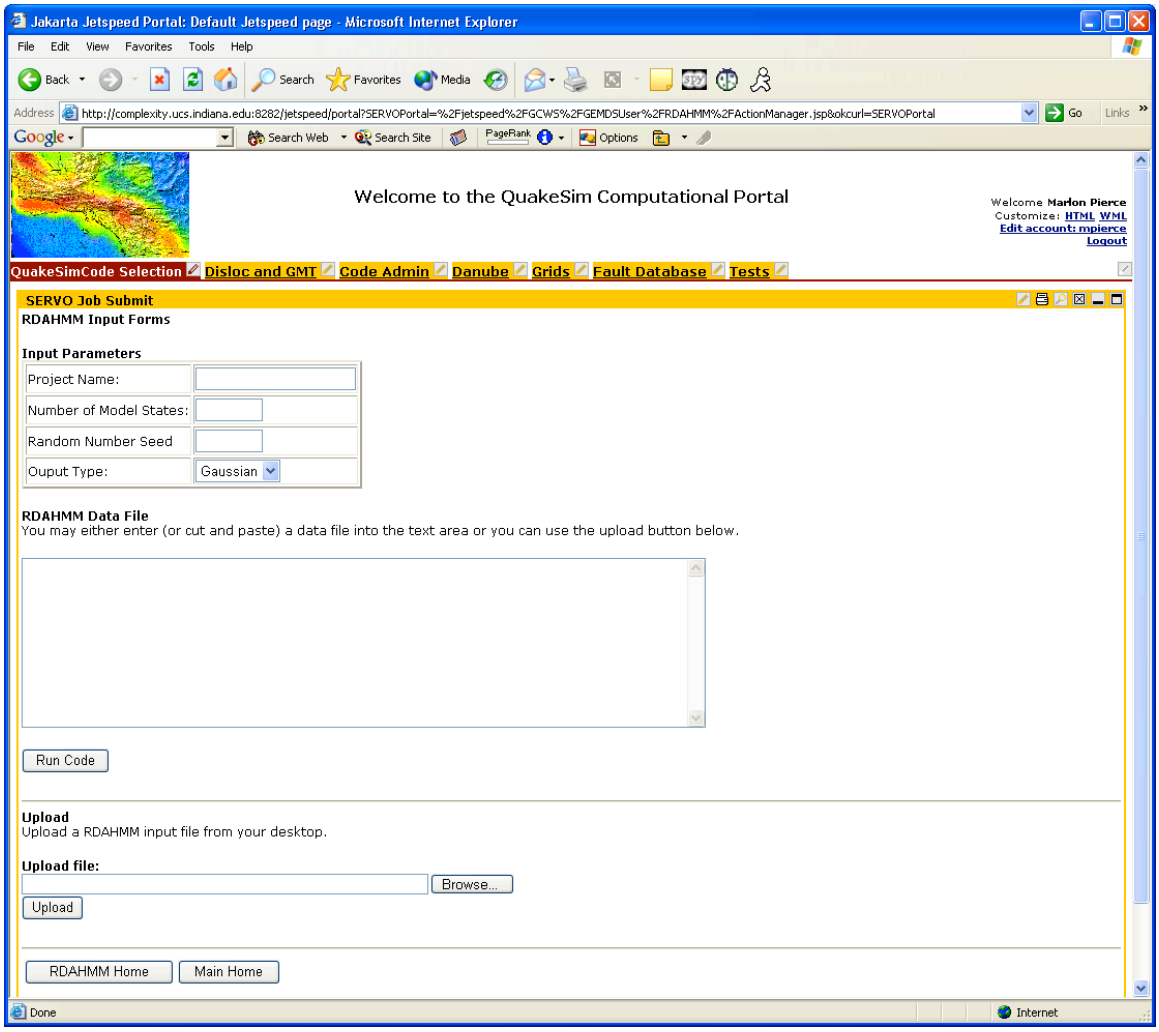

Figure 31 Form for running RDAHMM

Forms shown in Figure 32 require predefined input files created typically by hand editing of GPS or earthquake catalog data. The QuakeSim portal includes services for accessing and searching these resources online, as described in "Using the GPS and Seismic Catalog Databases" section below. Client components for accessing these data services have been integrated with RDAHMM.

To use the database for creating RDAHMM input files, you should first select "DB Project" in the RDAHMM main menu, Figure 31. Selecting this will lead you through steps illustrated in "Using the GPS and Seismic Catalog Databases" below. After you have made your selections and queried the database, your query results will be formatted appropriately for RDAHMM input, Figure 33.

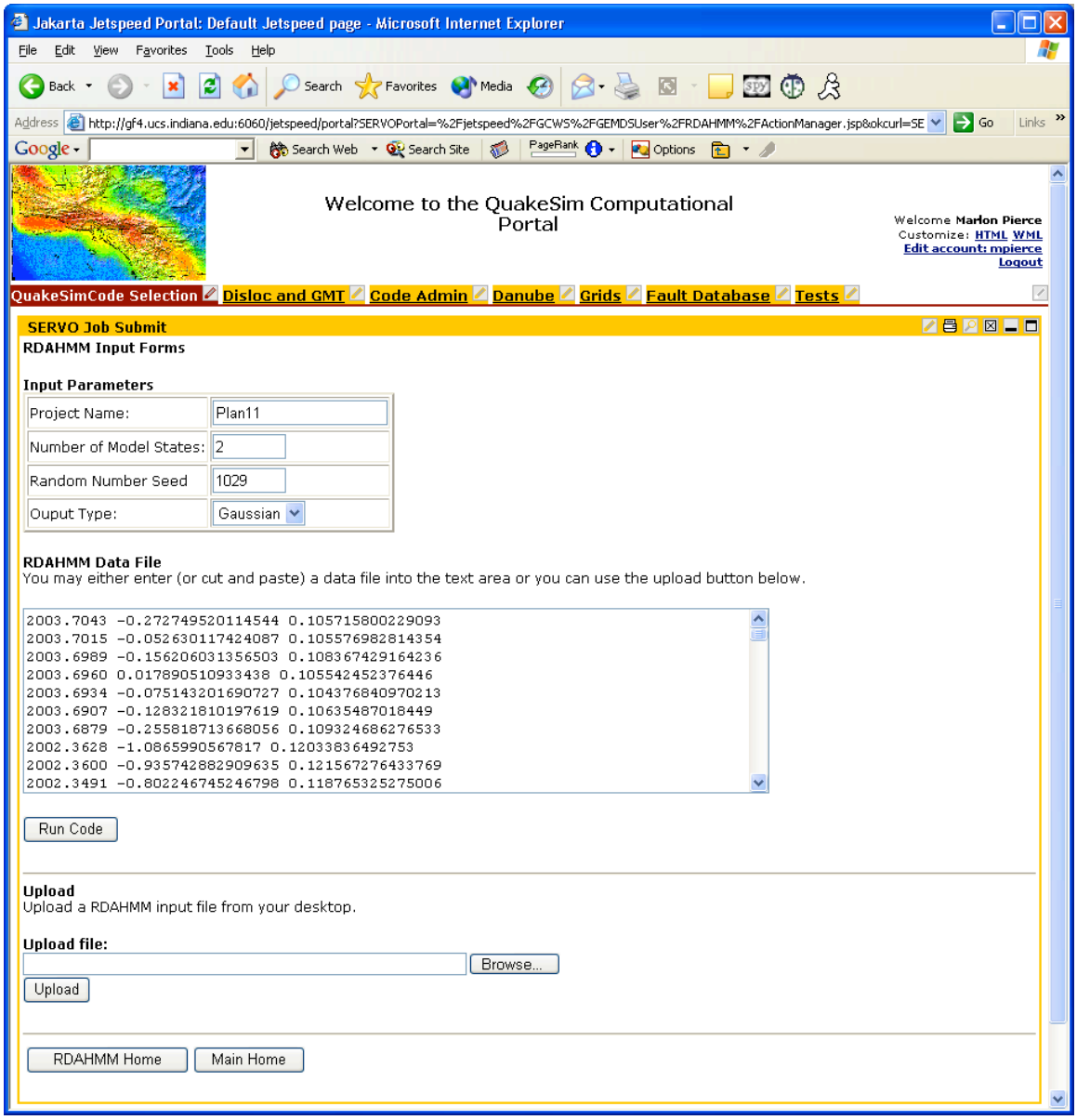

Figure 32 An RDAHMM submission page with data obtained from the GPS database

Note RDAHMM assumes all query fields are floating points, so some database fields (such as event quality) have no meaning and are excluded from the results. Other query results (specifically, date, latitude, and longitude) will be reformatted into decimal numbers.

Input files used by RDAHMM (either uploaded or obtained from the DB) are archived and may be recovered using the “Load Project” button on the RDAHMM main menu, as usual.

Running the Pattern Informatics Application

The Pattern Informatics application currently examines specified Seismic catalogs. This is not currently integrated with the Seismic Catalogs (as is RDAHMM). Sample values are provided in the input form, as is a default catalog. See Figure 32.

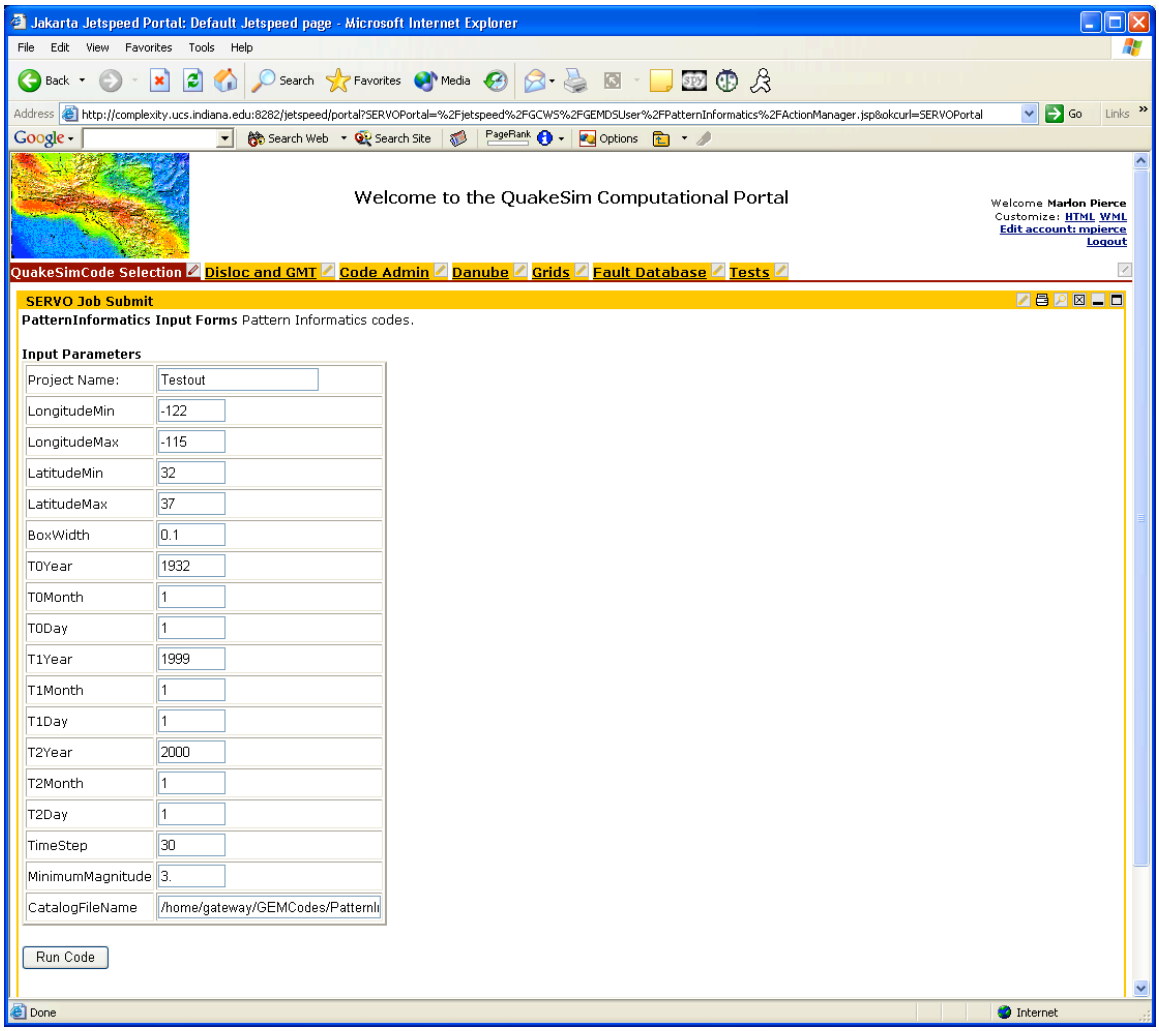

Figure 33 The Pattern Informatics form

After running the PI application, you may download the results in the usual manner (Figure 33) or you may use these results to generate a probability map, as shown in Figure 35.

TO DO: This part of the portal would be much better if integrated with the GPS and Seismic databases (as is RDAHMM).

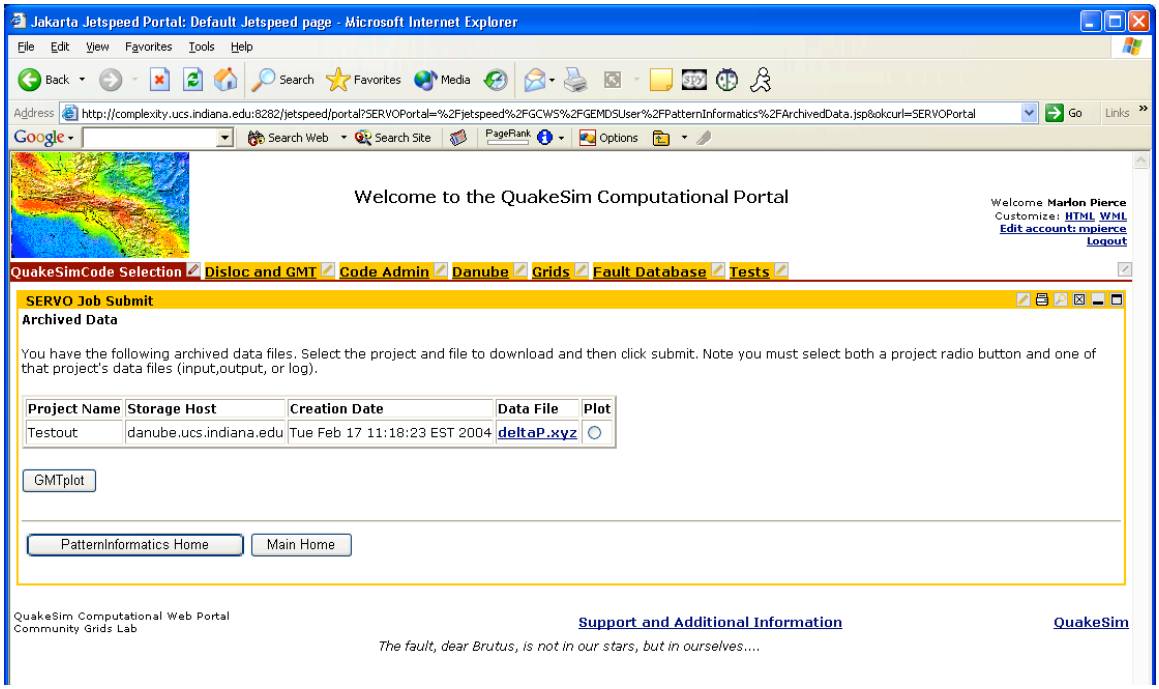

Figure 34 Download PI results or generate an image

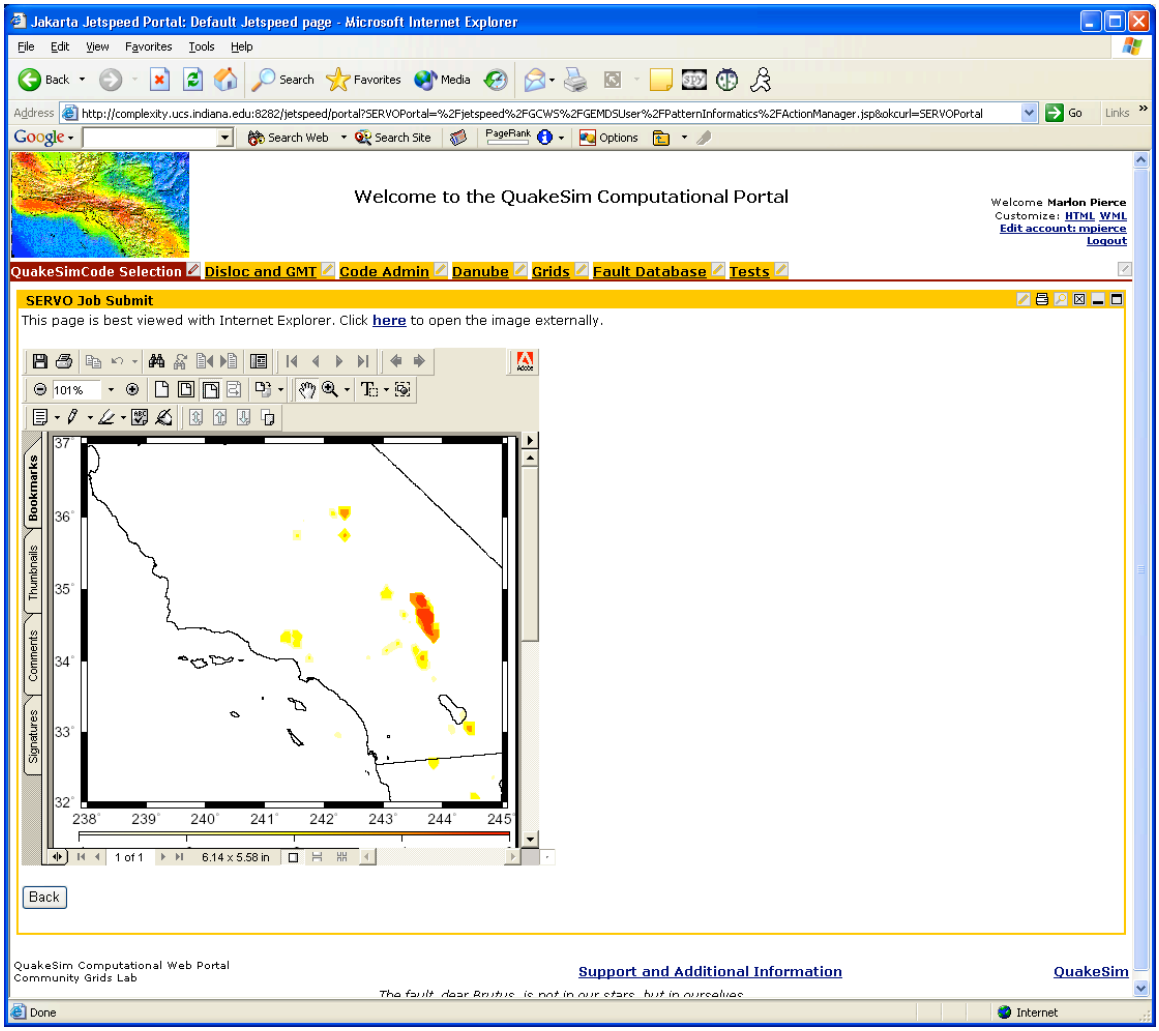

Figure 35 Sample GMT image of PI results

Running the Virtual California Codes

Virtual California (VC) is a suite of codes that must be executed in a particular order and require some user inspection, so before presenting the portal interface, some background is needed. More information on VC is available from http://www.servogrid.org/slide/GEM/VirtualCalifornia/README_VC.html. We do not explain the code physics here.

To run VC, the user must supply a fault topology file and a fault friction file. The fault topology file may be constructed (for example) from the QuakeTables fault database, although this functionality is not currently provided. Construction of the friction file is not covered here. Sample values of these files are provided in order to run the codes.

The steps to run VC are depicted in Figure 37. The first three codes run in sequence: VC_Stress_Green and VC_SG_Compress use the fault topology and friction files to create the stress Green's function. VC_INIT_SER is then run to thermalize the system. This code should be run repeatedly until all energy eigenvalues are negative. Finally, the actual earthquake simulation code, VC_SER, can be run. This should be done repeatedly, with the user inspecting output files to determine when the system has reached steady state.

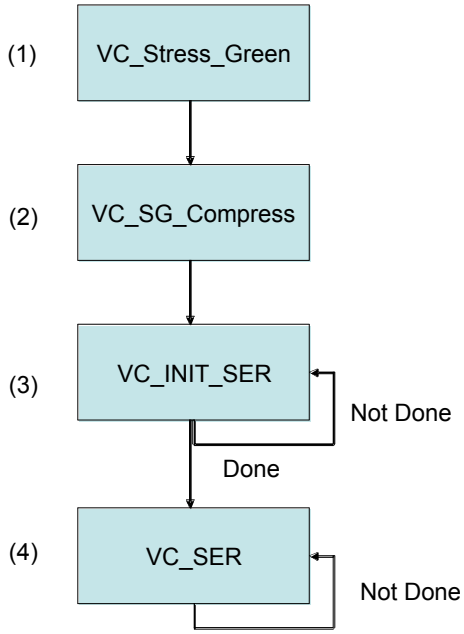

Figure 36 Simplified VC Flow Diagram

Visualization tools for VC outputs are available and will be included in future versions of the portal interface.

To use the VC interface, log into the portal as usual. From the Code Selection Menu, select VC and the desired host (danube in the test bed). The VC main page is shown in Figure 38. This differs from the usual code main page.

To start a new project, first select the "New Project" button. Provide a name for your project where prompted. You will then see a screen similar to Figure 39.

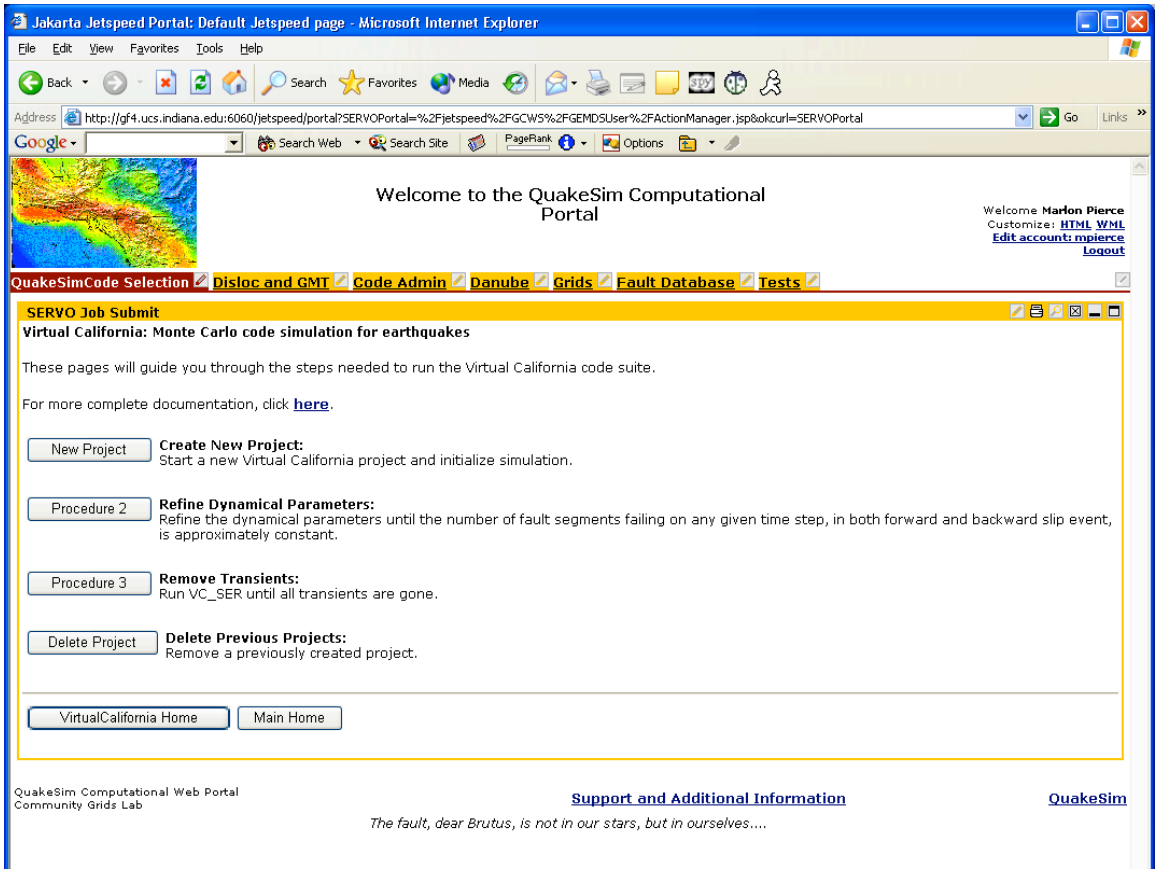

Figure 37 VC Main Page

Enter the requested information in the fields and click “Make Selection”. This will launch the initialization steps 1-3 of Figure 37. This will take several minutes to complete. You can follow the progress through the Job Monitor (under the Danube tab on the test bed interface).

When the initialization applications have finished, you may go to Procedure 2 in Figure 38. You should see a screen similar to Figure 40. The just-launched project initialization will not show up for several minutes in the project list, so if your new project is not in the list, it has not completed running.

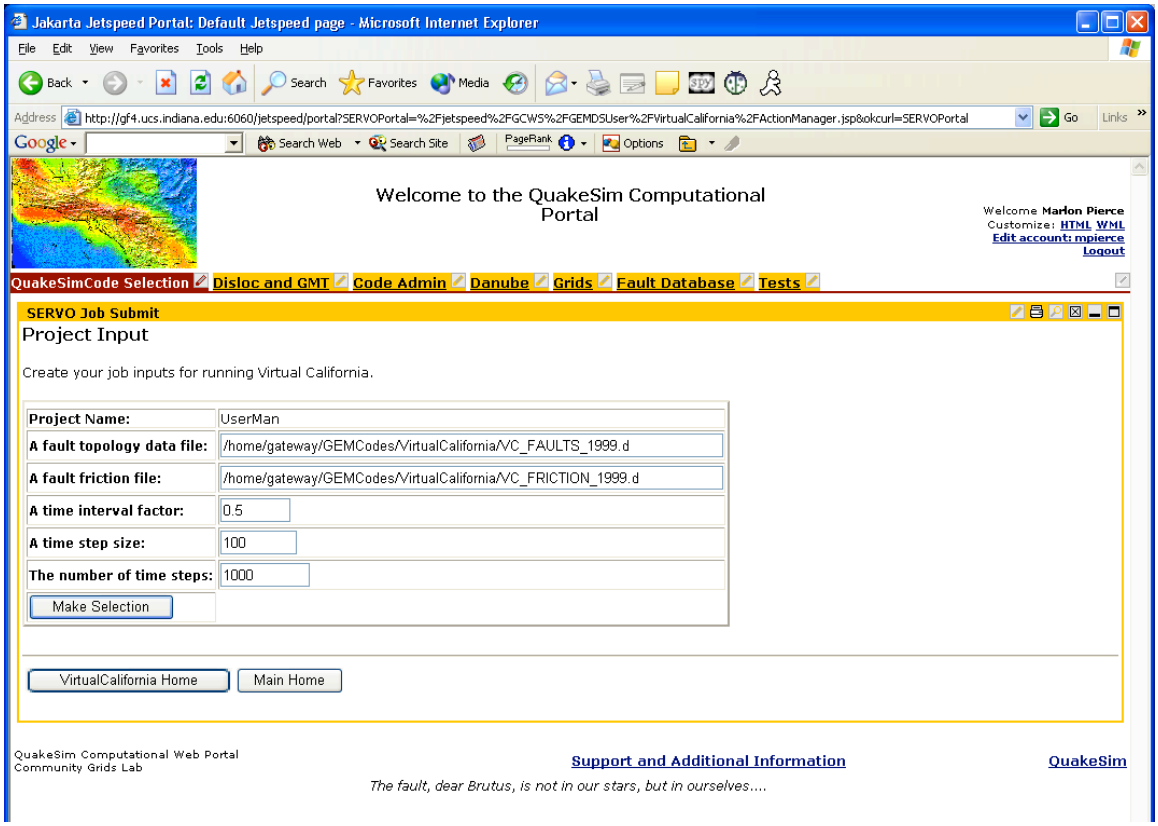

Figure 38 Setting up a new project

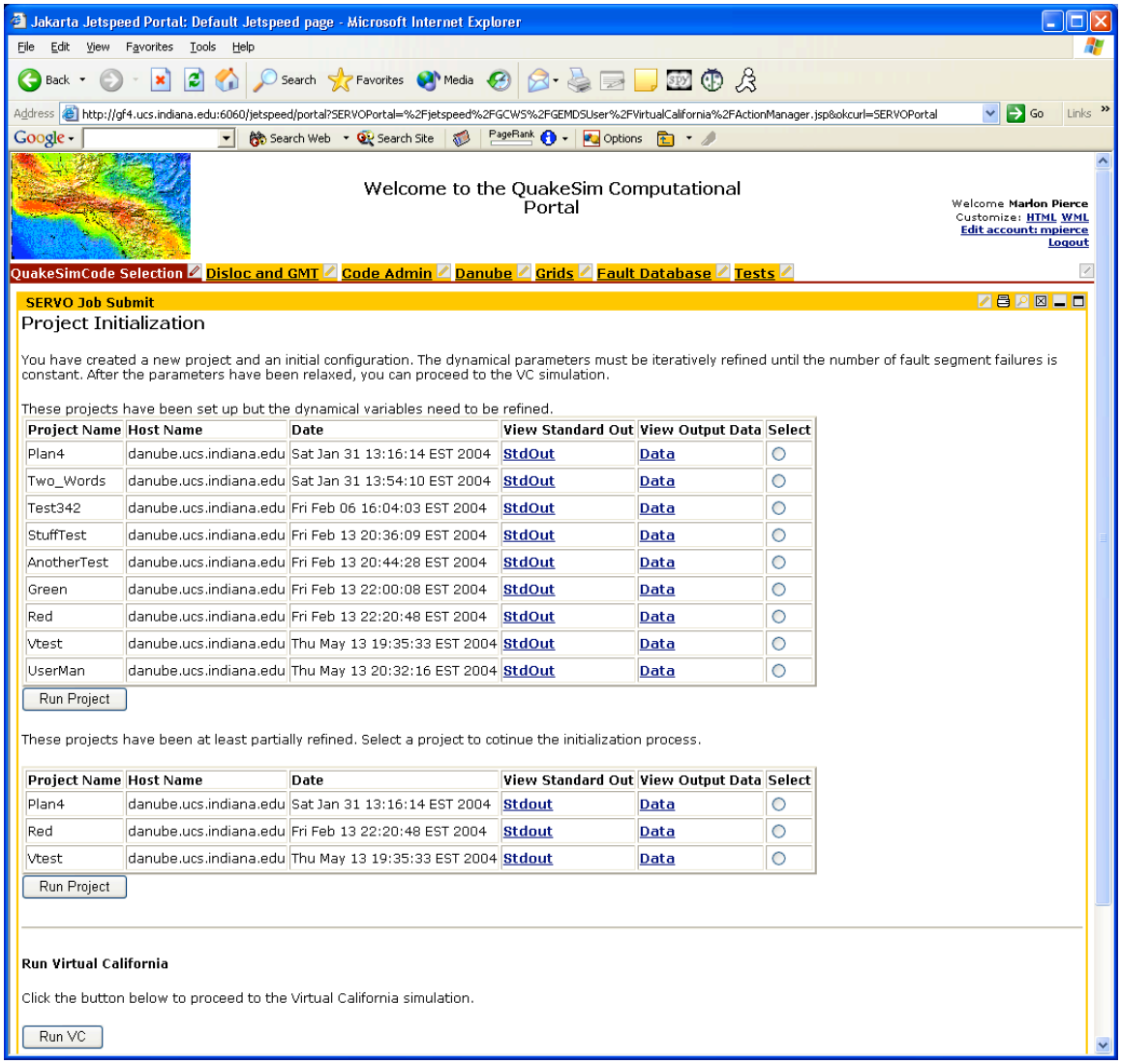

Figure 39 Screen shot of Procedure 2 (for running VC_INIT_SER)

The screen is split into two tables: the first consists of projects that have been created with steps 1-3 in Figure 38, but which have not yet been at least partially refined. The second table consists of projects that have been at least partially refined; that is, they are entries from the top table that have been refined at least once. You may view the standard output and the results file by clicking the links to the project.

Projects may appear in both lists. If you want to start a project over, select it from the top table. It will replace the entry in the lower table.

To refine an initialized project, select the radio button on the far right for the project entry and then select “Run Project” below the table. This will take you to an interface similar to Figure 41. Fill in the requested information and click “Make Selection”. This will launch VC_INIT_SER. Return to the Project Initialization window (Figure 40) and inspect the output. You should see that your project has been added to the lower table.

Select the code from this list and run until the VC dynamical parameters are sufficiently refined, as described in the VC user guide.

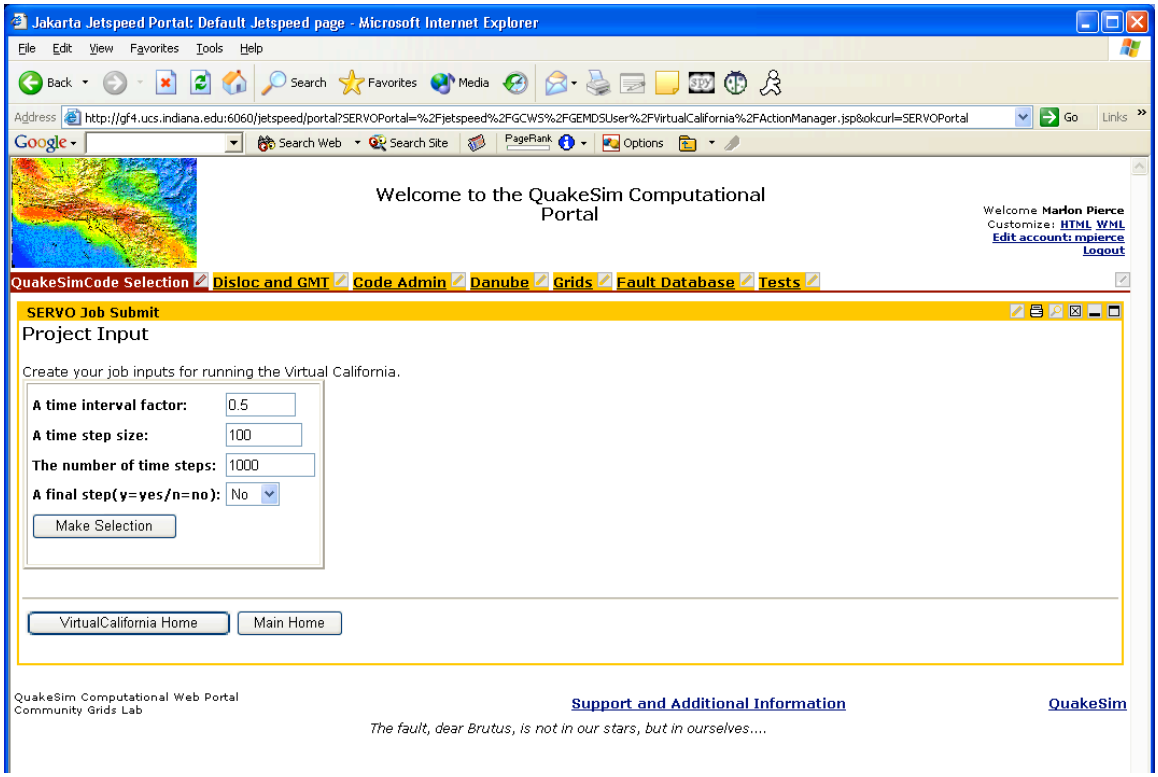

Figure 40 VC_INIT_SER input parameters

When you have sufficiently refined your input file, select Procedure 3, “Remove Transients” to run VC_SER. This will take you to a screen similar to Figure 41. The VC_SER page is again split into two tables: initialized and refined projects that have not yet been through the Monte Carlo runs, and projects that have been through at least one Monte Carlo run. Projects that have been through at least one run will be listed in both tables, so you can start over from the initialized data if you wish.

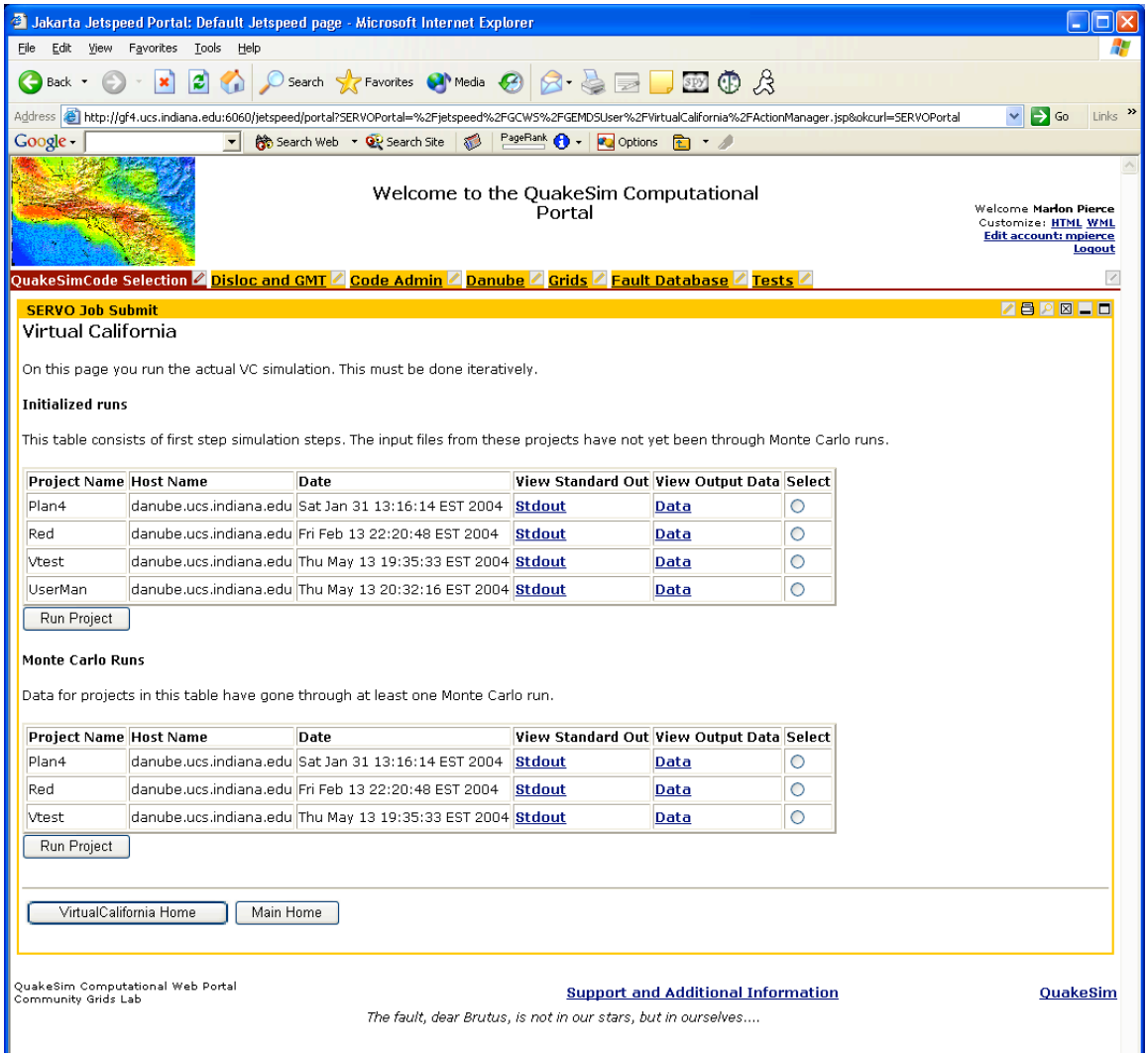

Figure 41 VC_SER menu page

You should continue to run Monte Carlo steps until the code output has reached a steady state.

Running Slider

Slider (a slider block model code) runs through the standard style user interface, similar to Figure 9. The portal interface follows the usage pattern for Disloc. See the Slider user guide for guidance with the input parameters.

Running the Genetic Algorithm Simulation

The Genetic Algorithm simulation runs through the standard style user interface, similar to Figure 9. The portal interface follows the usage pattern for Disloc. See the GA user guide for guidance with the input parameters.

Running Karhunen-Loeve

The Karhunen-Loeve code runs through the standard style user interface, similar to Figure 9. The portal interface follows the usage pattern for Disloc. See the K-L user guide for guidance with the input parameters.

Using the GPS and Seismic Catalog Databases

Introduction

The QuakeSim portal includes portlets for accessing GPS and Seismic catalogs. QuakeSim services can be used to harvest various formatted data files from public download sites, convert them into unified GML (Geography Markup Language) formats, and support SQL search filters on the data. These client interfaces are integrated with the RDAHMM application. Several others applications in the QuakeSim portfolio may be integrated in the future.

The following Seismic Catalog formats are supported:

1. SCSN
2. SCEDC
3. Dinger-Shearer
4. Haukkson

Current earthquake catalog sources include

1. http://www.data.scec.org:3128/ftp/catalogs/SCSN
2. http://www.data.scec.org:3128/ftp/catalogs/SCEC_DC

We also support the following GPS formats:

1. JPL
2. SOPAC
3. USGS

Current sources for GPS data harvesting are

1. ftp://sideshow.jpl.nasa.gov/pub/mbh/filtered
2. ftp://sideshow.jps.nasa.gov/pub/mbh/point

For detailed information, see http://grids.ucs.indiana.edu/~gaydin/servo/.

Portlet interfaces allow users to perform the following functions:

1. Download catalogs listed above
2. Convert them into GML formatted data.
3. Insert converted catalogs into a database
4. Perform searches on the database and download results.
5. View a list of downloaded catalogs
6. Manage (delete) entries in the database.

For demonstration purposes, all of these capabilities are currently available for all users: they are included in the template "gateway" account and will be added to all new user accounts. In future deployment, obviously most of these portlets (with the exception of the "search" interface) will be accessible only to privileged users. This security role system will be implemented using standard Jetspeed role definitions.

Performing Searches

First, log into the portal as normal. You should see a screen similar to Figure 4. Click the top level tab labeled "CCE DB" to take you to the screen shown in Figure 43. The portlet capabilities listed above are sidebar menu links on the left in the figure. Click "Search Catalogs" and then pick either "Search Earthquake Catalogs" or "Search GPS Catalogs." The rest of the instructions will assume the former option, but the GPS track is similar.

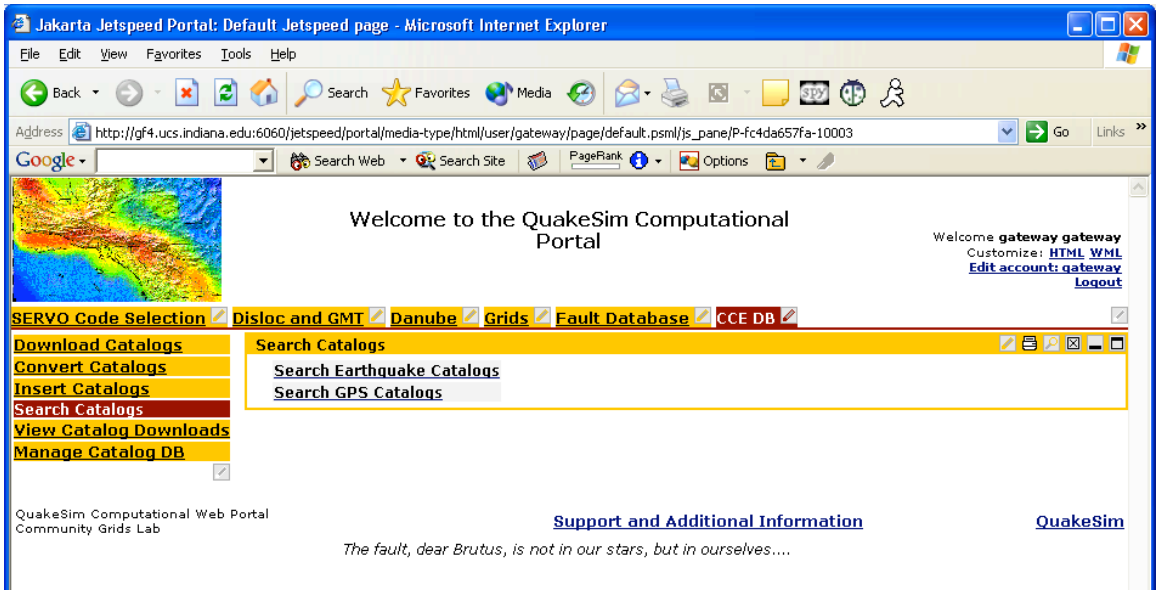

Figure 42 CCE Database navigation page

After clicking this link, you will see Figure 44. Select either SCSN or SCEDC format. Selecting SCSC will give you the screen shown in Figure 45. You may deselect (uncheck) any of the search fields that do not interest you.

After making your checkbox selections, a form similar to Figure 46 will be generated. Fill in all or part of the form. It is not necessary to complete the form: the default operation will select only fields with parameter values to refine this query. The screen shot shows (as an example) a request for all entries between 1933 and 1950 with a magnitude greater than 4.

Note: *Search results can be very large and query requests may time out for excessively large results. You should therefore always provide some limits on the queries you pose*.

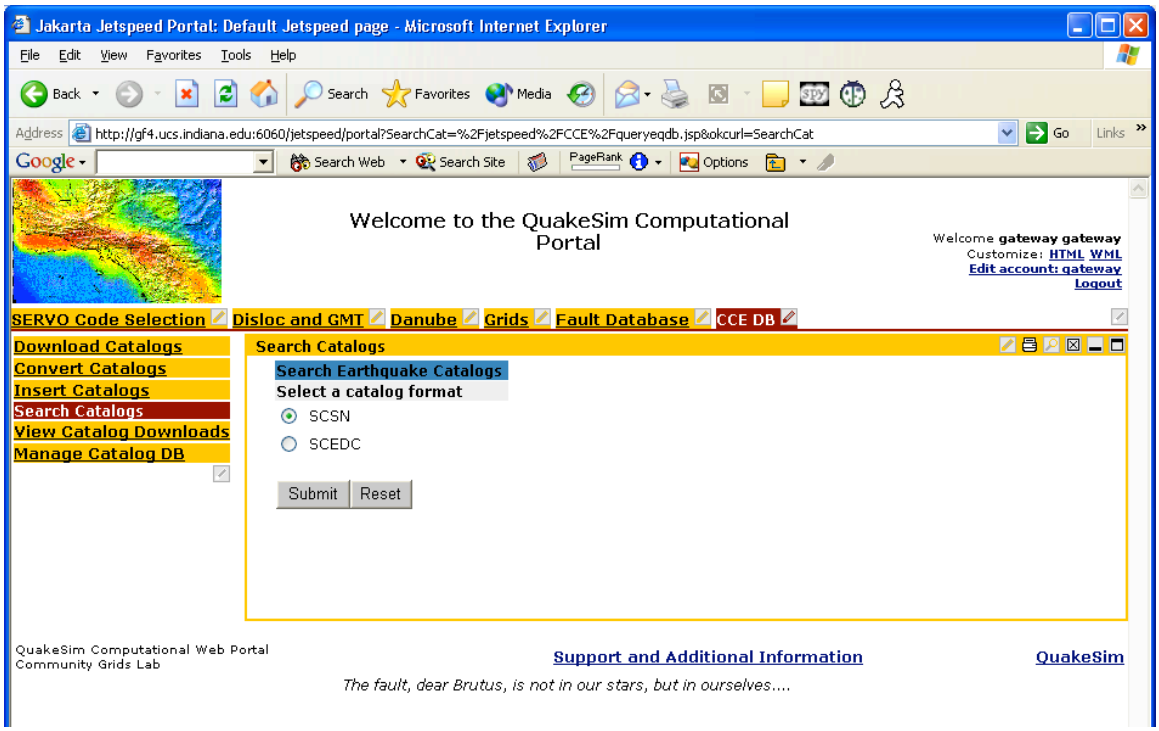

Figure 43 Select either SCSN or SCEDC formatted sources

The results of your search will be displayed in the portal window (as shown in Figure 47). You may download these results into a separate browser window by clicking the "Download Results" link. You may also create a "Printer Friendly" (and download friendly) version of the display by clicking the small printer icon in the upper right hand of the portlet control bar. After clicking this icon, you should see a screen similar to Figure 48. Note this does not open a separate browser window, so to go back to the portal, click the "Back" button on your browser.

To go back to the main search menu, click the "Start Over" button at the bottom of the results. If your results are too long to be displayed in the window (as in Figure 47), scroll down to the bottom of the page with the right hand slider bar.

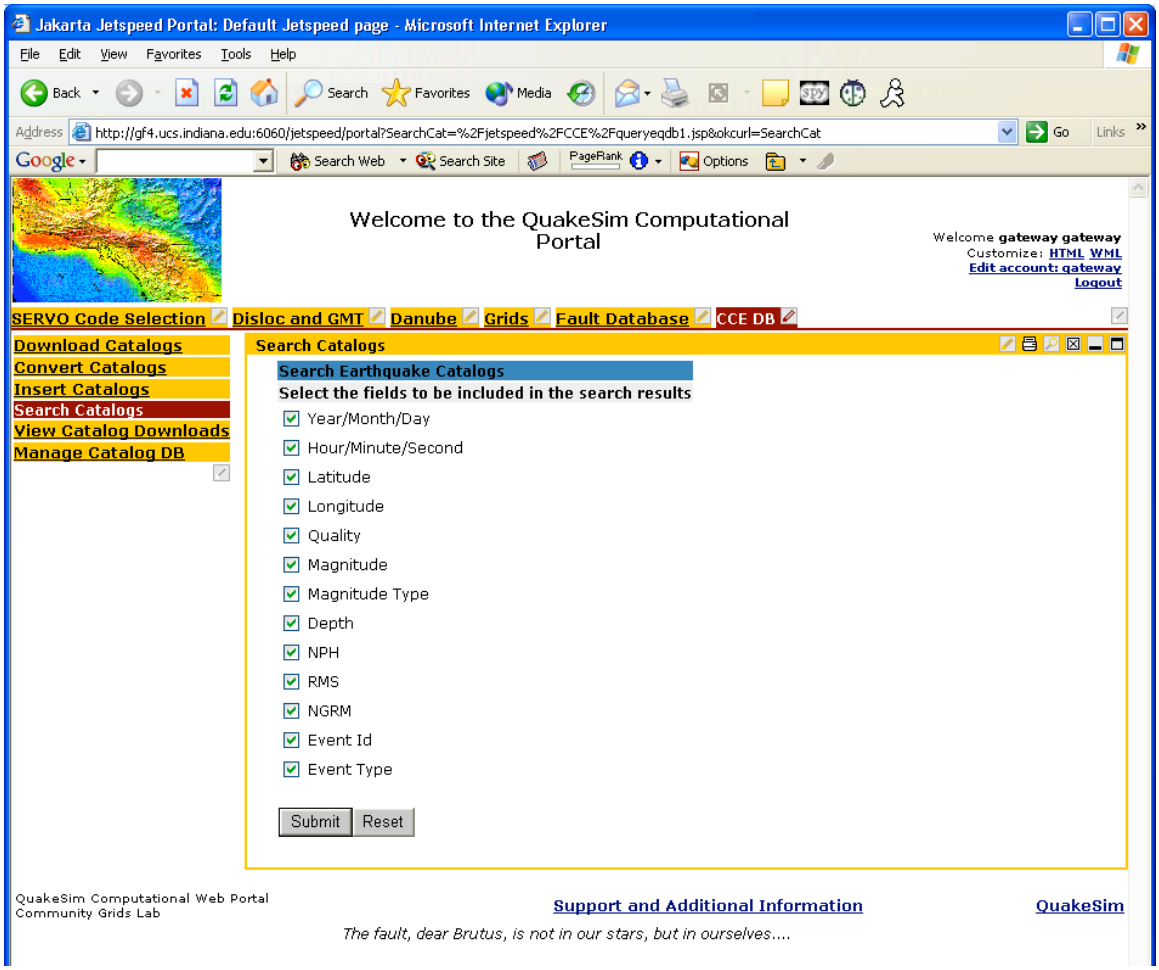

Figure 44 Select the SCSN fields you want to search over

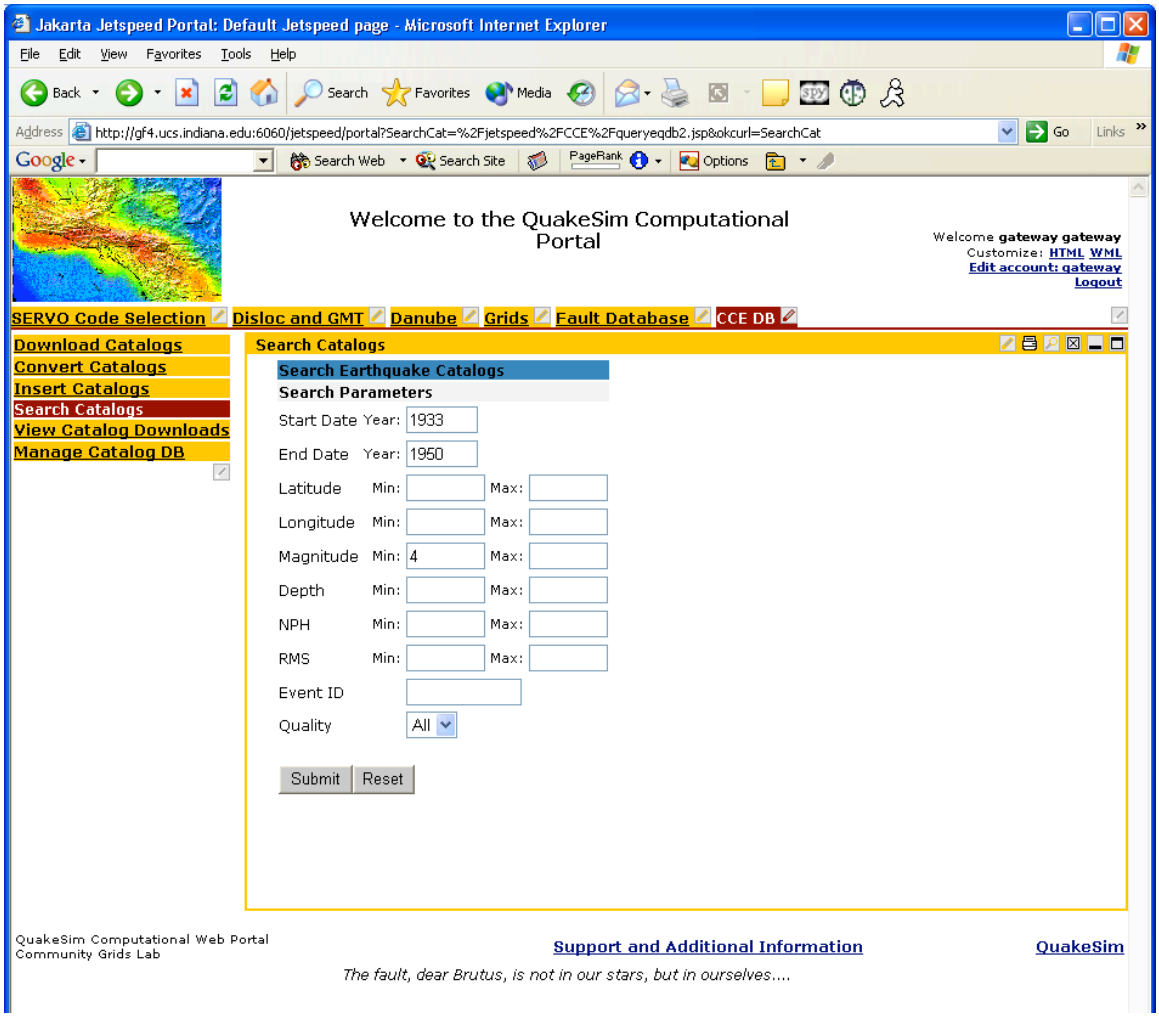

Figure 45 SCSN formatted data search parameter forms

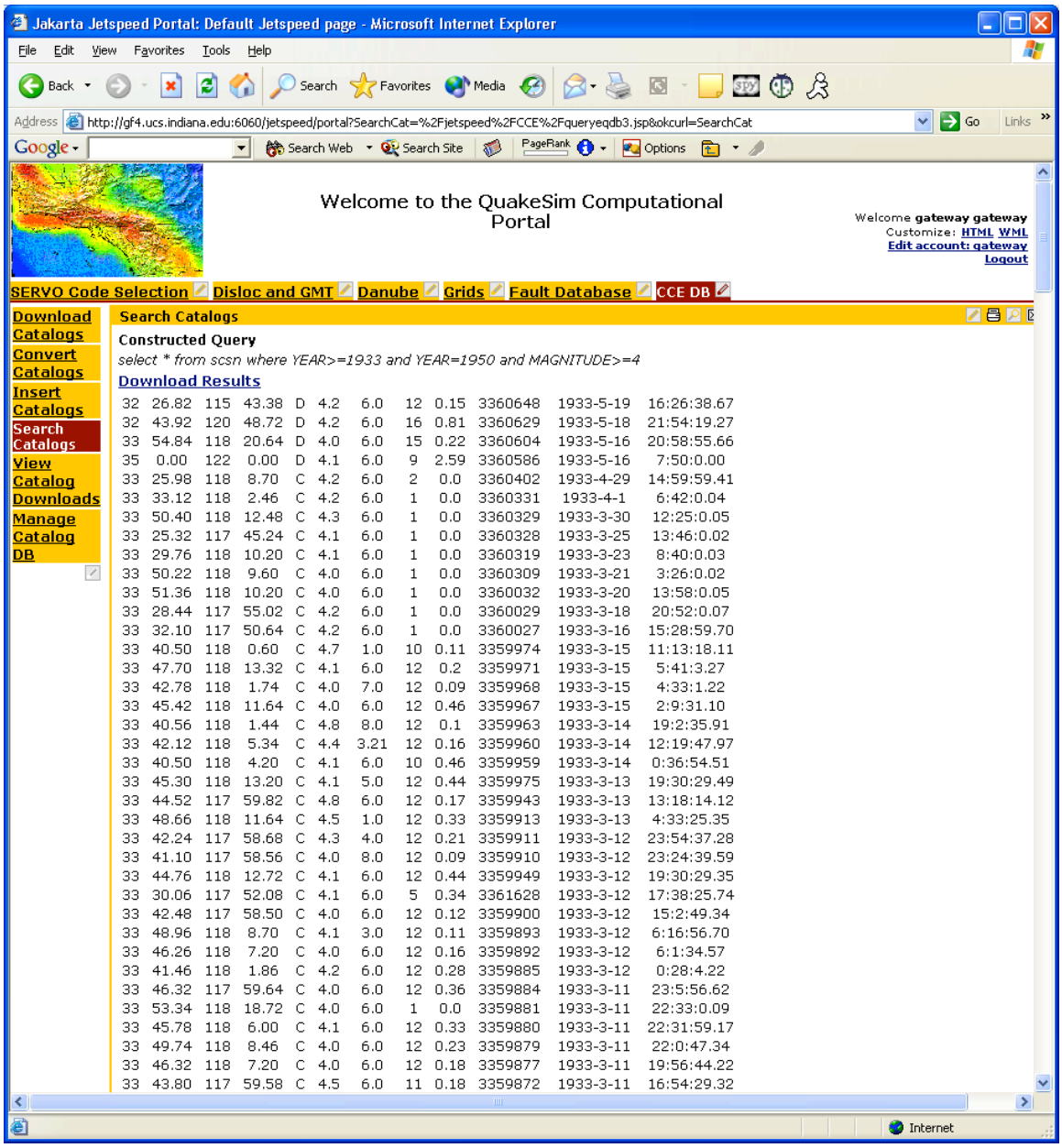

Figure 46 Search results are displayed and may be downloaded

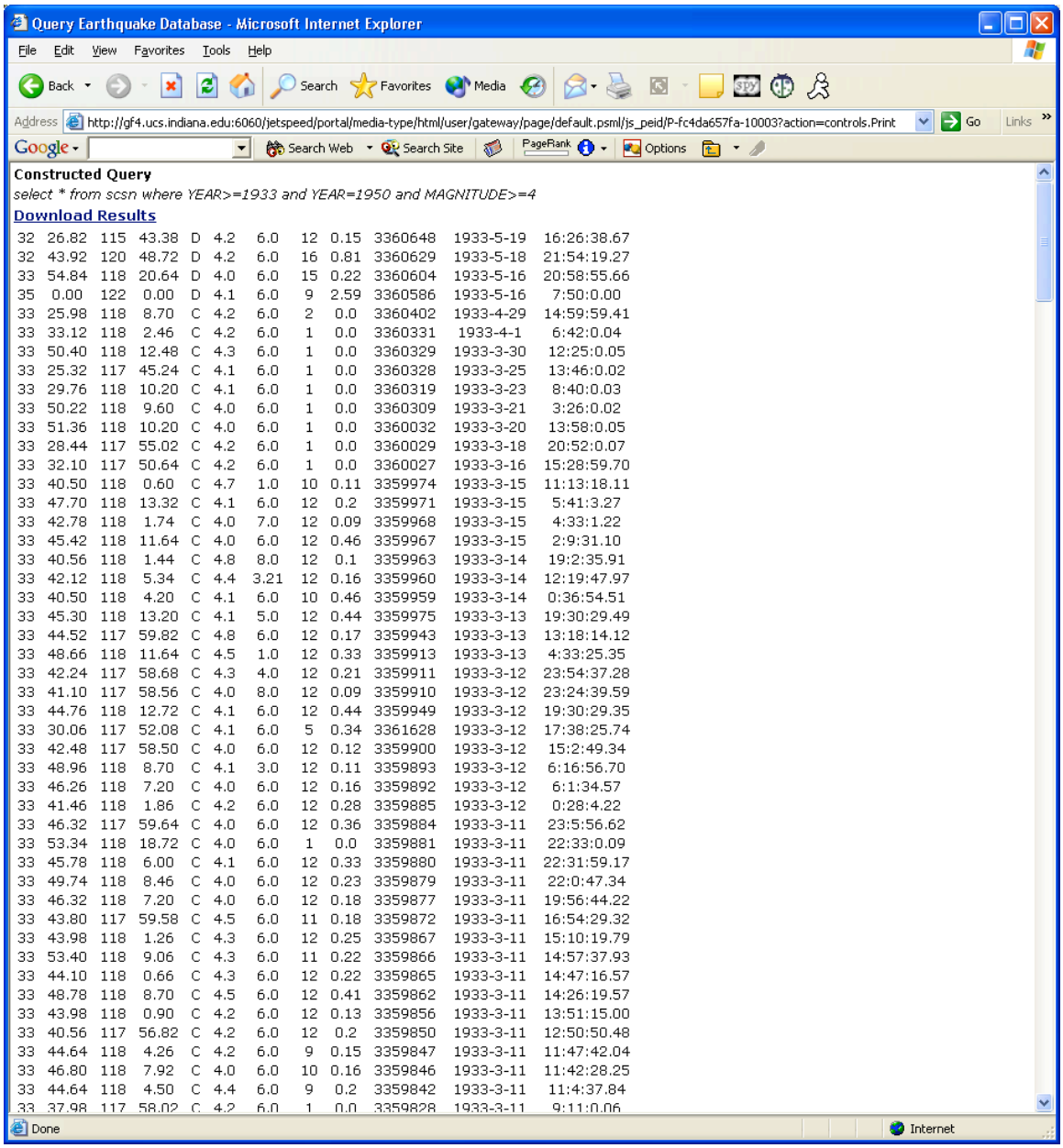

Figure 47 Printer friendly display of the search results

Note on Browser "Back" Buttons

Many of the portal pages are dynamically generated from HTTP request parameters passed from the previous page. As a consequence, if you click the "Back" button on your browser, you will often see a screen similar to Figure 49.

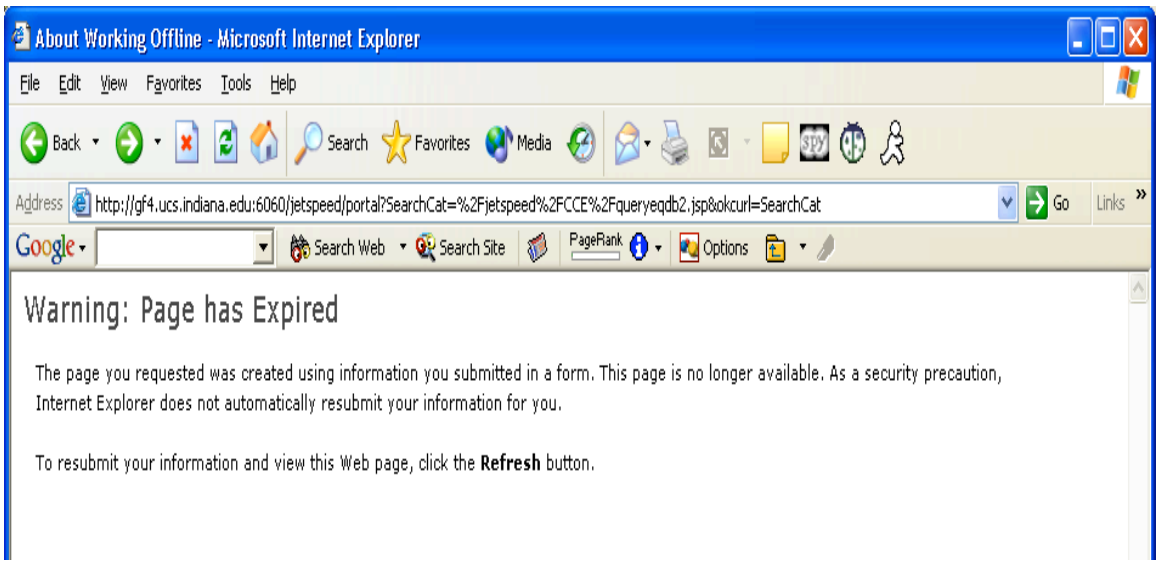

Figure 48 Expired pages may be reloaded with the browser refresh button

If you encounter this page, you need to simply click the “Refresh” (or “Reload”) button on your browser. When prompted in the popup window, click “Retry” to resend any stored HTTP parameters. After this, the desired page should be reloaded.

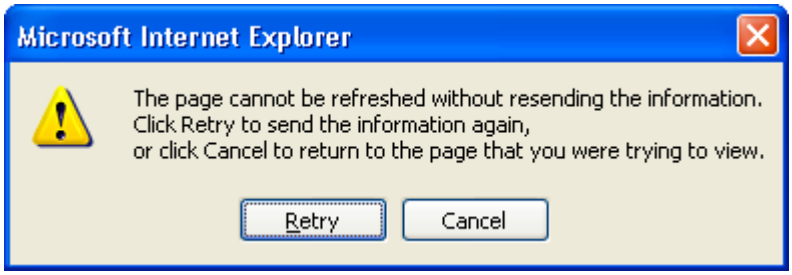

Figure 49 After clicking the refresh window, click "Retry" in the popup

Troubleshooting Problems

- If you have problems or wish to report bugs, please contact Marlon Pierce: marpierc@indiana.edu or 812-856-1212. You may also contact Choonhan Youn, cyoun@ecs.syr.edu if the servers need to be restarted.
- The portal is just beginning user testing and bugs will be found, so if you find a problem that you can’t recover from, close your browser and open a new one. This will remove all browser sessions.
- Remember that the portal currently only supports meshes created from 3 layer geometries with a single embedded fault.

Third Party Codes and GeoFEST Helper Applications

The portal is based primarily on the following technologies:

- Jetspeed is used to build the portal framework (for which we have added some extensions).
- JavaServer Pages are used to create the web pages.
- Remote invocations and communications between various machines and databases in the test bed use Web services running in the Apache Axis implementation of WSDL and SOAP.
- We use apache-ant (with various custom extensions) to coordinate related tasks (such as the steps needed to create the initial mesh or to generate the movie).

The mesh generation, refinement, input creation and movie creation are all done through calls to several external programs. We typically use apache-ant to invoke these non-Java applications.

- AKIRA is used to generate the mesh. We call Akira directly and also through a legacy control code, CROM.
- APOLLO is used for mesh refinement (through tagfault.pl).
- GeoFEST uses a number of helper Perl scripts (refonce.pl, tagfault.pl, and lee2geof43m) to create the mesh and apply boundary conditions and material properties.
- The mpegs are generated using RIVA.

An earlier graphical interface for AKIRA, CROM and APOLLO (the mesh generation technology used in this portal) is described in the Appendix of the Milestone I Portal Example documentation:

http://www-aig.jpl.nasa.gov/public/dus/quakesim/PortalExample.pdf.